E.T.S. de Ingeniería Industrial,
Informatica y de Telecomunicación

ANSYS Forte Simulate Guide

Grado en Ingeniería

en Tecnologías Industriales

Trabajo Fin de Grado

Sergio Pérez de Albéniz Azqueta

Pedro María Dieguez Elizondo

Pamplona, 10/06/2020

ACKNOWLEDGEMENTS

I would like to thank this final project to everyone that has been supporting me during all this tough years, despite of the hard issues, I am grateful to all this amazing people that made this period the most enjoyable of my life.

First of all, I would like to highlight my family back up, because no matter what happen or what goes wrong they have been always my strength to achieve anything in my live. I would not imagine how I could go through this years without them. I will always remember my parent's advices, because it have been so helpful during my live. How can I forgive my sister's support, always trying to help and encourage her little son, my aunt's sympathy and smiling face that life up in the difficult moments, and last but not least the love of my grandmother that has been in my heart since i was i child protecting me.

Also I appreciate my friend's group support, whose most of them I grew up since we were children. Most of my time I spent with them because they deserve it. I have no words to drive the emotional connections I have with them, due to all the exprimes we have been through together. They become my family, my brothers, along my life and I always been in debt with them. I will always appreciate their support and comprehension.

Finally, I would like to express my gratitud with every single teacher I had since I was a kid, because if they would not guide my I would not end where I am now.

ABSTRACT

The purpose of the current projects is the first introduction of the Computer Aided Engineering (CAE) software ANSYS Forte to the Public University of Navarra.

ANSYS Forte is specializes in simulating processes for an internal combustion engines and it is divide in three components: Simulate, Monitor and Visualize, each options gives special access in order to the intention.

The main goal for this final project is witting a guide for the use of ANSYS Forte Simulate. Because it is the beginning of the simulation process, where to set up and execute the model. Along the report it is explained the commands needed for the configuration of an engine in terms of mesh, chemistry data, geometry and model parameters. The present project involves different areas of engineering such as: mechanics, thermal engines, computing and design.

KEY WORDS

- ANSYS FORTE
- SIMULATE
- THERMAL ENGINE
- UNDIREC INJECTED
- SPARK IGNITION

INDEX

FIGURES INDEX

TABLES INDEX

1. Introduction

Finite element method (FEM) for solving partial differential equations is used by many softwares. ANSYS Forte fluid-dynamics is a Computer Aided Engineering (CAE) software specializes in simulating processes of combustion in an internal combustion engine, using a highly efficient combination of comprehensive chemical kinetics, liquid fuel spray and turbulent gas dynamics.

A simulation of ANSYS Forte solves the complete Reynolds-averaged Navier-Stokes (RANS) equations with well developed models of fluid turbulence. However, the ANSYS Forte CFD simulation program, distinguished from other CFD apps, employs sophisticated spray models that drastically lower grid resolution specifications and simulation time-step dependencies.

In addition, ANSYS Forte software introduces an innovative, advanced chemical solver module. This capacity makes direct use of reaction processes that include hundreds of chemical compounds, with simulation periods typically associated with processes that are smaller order of magnitude.

In 3-D engine models, the integration of sophisticated chemical solver and multi-component fuel vaporization models opens the door to simulating practical fuel surrogates. It has the option to use automated, on-the-fly mesh creation or state-of-the-art mesh-moving algorithms with pre-generated body-fitted meshes while using such advanced fluid models.

Included in ANSYS Forte, a sector-mesh generator allows simulation of symmetric structures using the body-fitted mesh-movement method. Finally, the ANSYS Forte user interface offers a directed user environment for setting up, simulating, and visualizing results, in addition to the simulation capabilities.

1.1 Workflow

When ANSYS Forte first starts, a launch panel displays, giving the option for the three different options:

1. **Simulate** - Start setting up and running the model, including mesh and chemical data selection, engine and model parameter specification, job performance, and real-time run monitoring.

2. **Monitor** - When the job is running or after it has completed it is use to create plots.

3. **Visualize** - It is the launcher for the results. Visualizations include spatially average values of surface contour maps, vector graphs, droplet view, animations and X-Y line maps, point interpolations, and line interpolations.

2. ANSYS FORTE SIMULATE

The usage and features of ANSYS Forte Simulate will be explained alongside this tutorial, as they state before is the start to set up and run a project.

A workflow tree will appear in his left-hand panel until it is installed. The nodes in this tree are linked to the simulation's set-up and job execution tasks, the goal being to work from top to bottom. **Figure 2.1: ANSYS Forte Simulate: Workflow overview** provides a high-level view of the Simulation tasks.

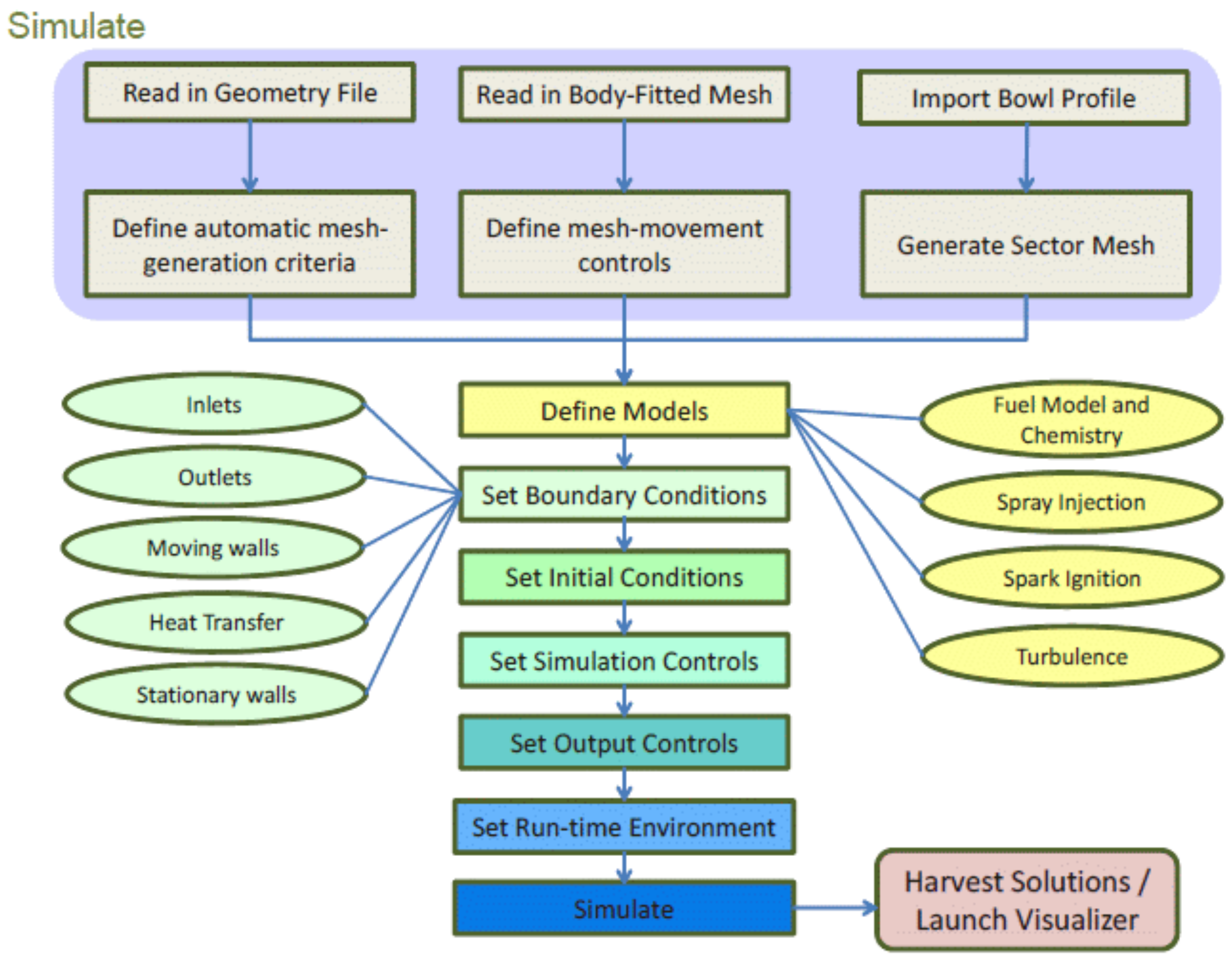

FIGURE 2.1: ANSYS FORTE SIMULATE: WORKFLOW OVERVIEW

The ANSYS Forte Simulate windows consist of sub-panels, a top-level menu bar, and a button-shortcuts to specific tasks to be used. Commands for general activities are placed across the top of the ANSYS Forte panel, in the menu bar open. Commands that are commonly used are accessible via shortcuts in the toolbar.

Figure 2.2: Layout of the ANSYS Forte Simulation window provides a map of the different areas inside the Simulate frame, offering easy access to the various tasks during project setup.

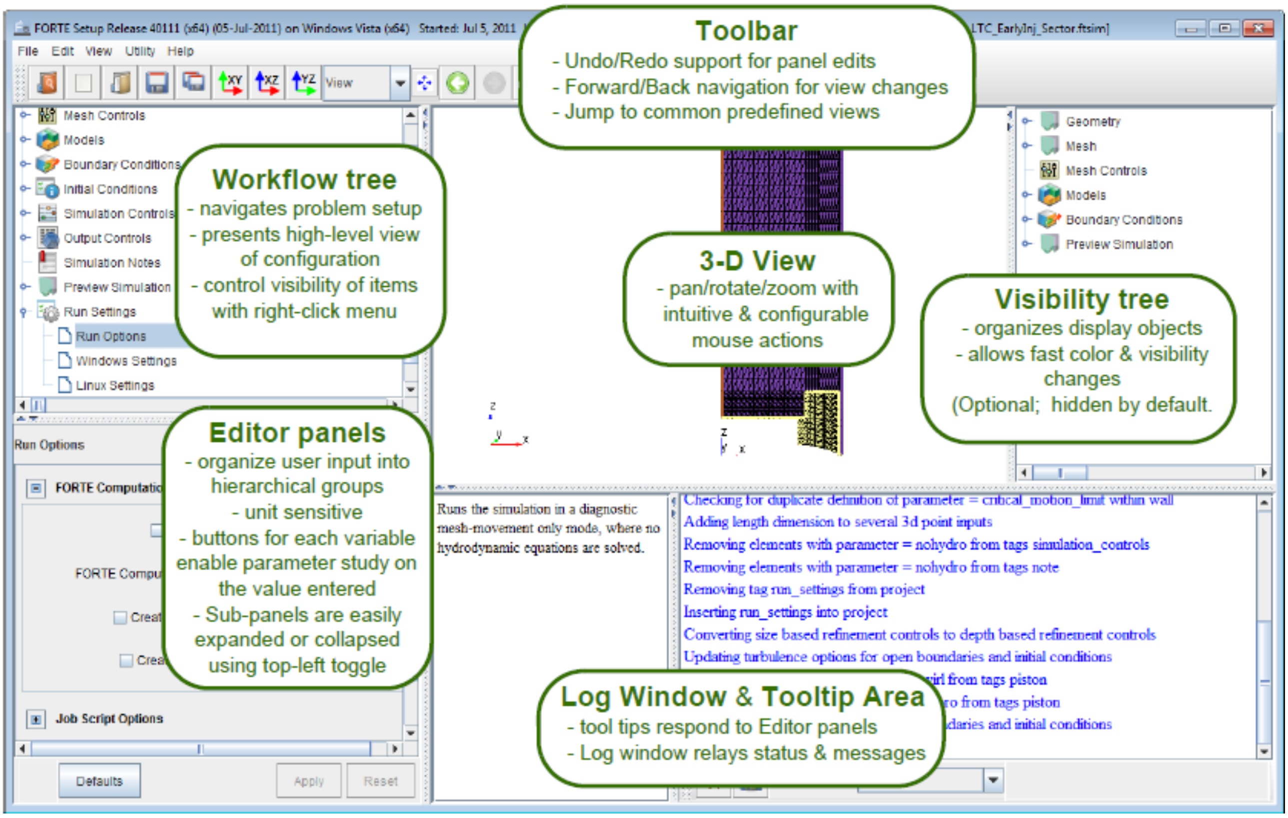

FIGURE 2.2: LAYOUT OF THE ANSYS FORTE SIMULATION WINDOW

2.1. Menus and Menu Items

Commands for general tasks are located in the menus accessible from the menu bar across the top of the ANSYS Forte window, where it is includes the top-level menus: File, Edit, View, Utility and Help. The following tables will describe the the tasks of each pull-down menu and the action that will occur when it is selected.

Table 2.1: File menu

Icon	Command	Description
	**Open...**	Opens an existing ANSYS Forte simulation file (***.ftsim**).
	**Save...**	Saves the currently opened ANSYS Forte simulation to a file (typically, ***.ftsim**).
	**Save As...**	Saves the current ANSYS Forte simulation to a new file with a user- supplied name.
	**Close...**	Closes the current project.
	**Exit**	Exits the current session and closes the window.

Table 2.2: Edit menu

Icon	Command	Description
	**Edit Preferences**	Launches an editor for setting various user preferences. See User Preferences , for more details on preferences.
	**Profiles...**	Launches the Profile Editor for creating profiles or editing profiles saved to the project. See Entering Profile Data and Profile Editor , for more details on editing profiles.

	**Mixtures...**	Launches the Mixture Editor for creating gas or fuel mixtures or editing mixtures or compositions saved to the project. See Mixture Editor , for more details on editing mixtures.
	**Dispersed Phases...**	Launches the Dispersed Phase Editor for creating surface composition and dispersed phase properties associated with particle tracking, and saved to the project. See Solid Phase Editor and Dispersed Phase Editor for Particle Tracking , for more details on editing solid phase properties.
	**Flame Tables...**	Launches the Flame Speed Table Editor for creating or importing tables of laminar flame speeds or editing flame speed tables saved to the project. See Flame- Speed Table Editor , for more details.
	**Initial Condition Tables...**	Launches the Initial Condition Table Editor for creating or importing tables of initial conditions or editing initial condition tables saved to the project.
	**Solution Criteria...**	Opens the Solution Criteria dialog to specify the steady-state conditions based on location and solution variables. See Solid Phase Editor and Dispersed Phase Editor for Particle Tracking , and Steady- State Simulation , for more details.
	**Boundary Condition Tables...**	Launches the Boundary Condition Table Editor for creating or importing tables of boundary conditions or editing boundary condition tables saved to the project.
	**Parameter Studies...**	Launches the Parameter Study Editor. See Parameter Studies , for more details.

Table 2.3: View menu

Icon	Command	Description
	XY	Orients the view to an XY projection.
	**XZ**	Orients the view to an XZ projection.
	**YZ**	Orients the view to a YZ projection.
	**Refit**	Centers the visible items in the 3-D view window and zooms in or out to encompass the entire visible geometry.
	**Previous View**	Returns to the previous view orientation.
	**Next View**	Restores the last view orientation since a **Previous View** action was performed.
	**Alter Visibility**	Launches a dialog used to alter the visibility of items by multi-selecting from a list or through screen selection. See Setting Mesh Display Attributes.

Table 2.4: Utility menu

Icon	Command	Description
	Pre-Processing	Displays a Pre-Processing panel that allows definition and verification of a chemistry set.
	Validate Project	Checks for inconsistent setup options and missing values.
	Open File Browser	Opens an operating-system file browser at the user directory.
	BDC =>TDC Transform	Runs the BDC to TDC Transform utility to shift the mesh to Top Dead Center by splitting the liner, shifting the moving surface, and stitching together open triangles. See also Geometry Node .

	Open Composition Calculator	Opens the Composition Calculator for specifying the initial (and boundary condition) composition information in ANSYS Forte for in-cylinder, exhaust-port and inlet-port initialization. See also Composition Calculator .

Table 2.5: Help menu

Command	Description
ANSYS Forte Help	Opens the entry page listing the ANSYS Forte Help. The default location is the ANSYS Help site
Release Notes	Opens to the current Forte Release Notes.
Best Practices	Opens to a document providing guidance on best practices in a variety of Forte simulations.
Forte Quick Start Guide	Opens the Forte Quick Start Guide, with introductory tutorial exercises to help you become acquainted with Forte.
Forte Theory Manual	Opens theANSYS Forte Theory Manual .
Forte Tutorials	Opens the Forte Tutorials, which provide example cases and access to the associated sample files.
ANSYS Forte User's Guide	Opens this ANSYS Forte User's Guide.
ANSYS Product Improvement Program	Provides link to information about this ANSYS quality program.
About FORTE	Provides version number, patch details, and other information about the installed software.

2.2. Workflow Tree

The Workflow tree is in the upper left corner of the ANSYS Forte browser. (see **Figure 2.2: Layout of the ANSYS Forte Simulation window**).

The workflow tree enables a fast navigation around the various nodes on the tree, these nodes represent the steps needed to set up and run a simulation or parameter analysis involving several simulations, and are ordered in a prescribed top-down workflow order, although any node can be picked and updated at any time.

In the Workflow tree a pull-down menu with several functions is available by right-clicking and choosing commands from the background menu. (see **Figure 2.3: Context-sensitive right-click menu for Chemistry node in Workflow tree**).

Once choosing an object, an Editor panel will be placed in the lower-left portion of the browser, where a list of links allows the items to access easily. as shown in **Figure 2.4: Links for navigating Workflow tree in Editor panels**.

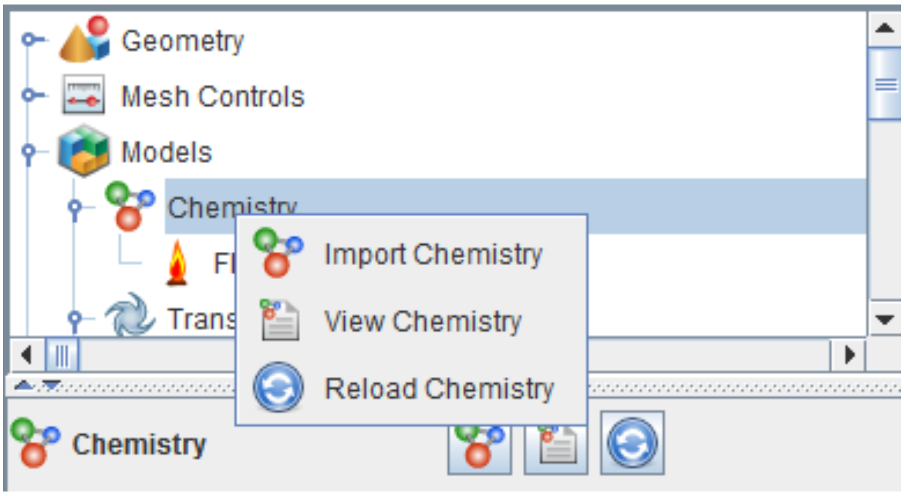

FIGURE 2.3: CONTEXT-SENSITIVE RIGHT-CLICK MENU FOR CHEMISTRY NODE IN WORKFLOW TREE

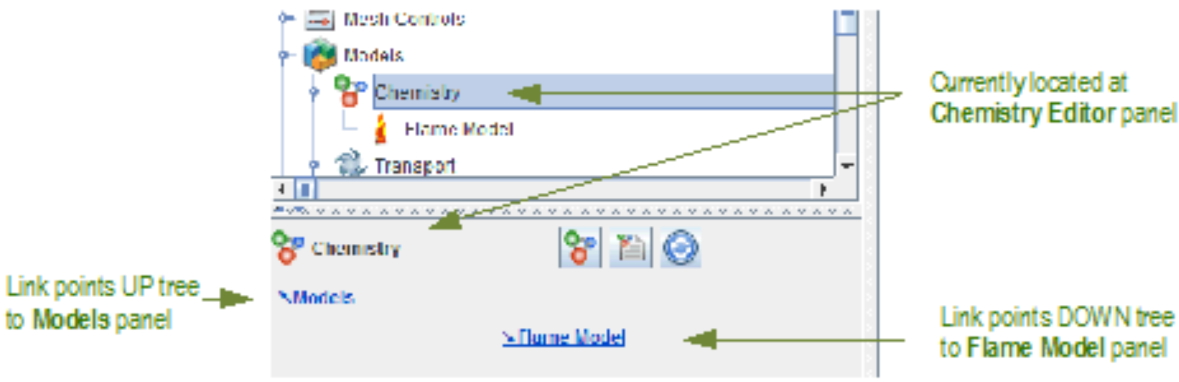

FIGURE 2.4: LINKS FOR NAVIGATING WORKFLOW TREE IN EDITOR PANELS

2.3. Editor Panels

Editor panels are shown in the lower left corner and each panel provides a range of similar configurations or choices to be determined based on the parameters of the platform. Those panels are not usable until they are turned on. (checkmarked).

Each time any adjustment is made in any Editor panel, the Apply button must have been used before exiting the panel for the inputs to take effect.

2.3.1. Standard Buttons in Editor Panels

Every Editor Panel displays a set of standard buttons at the bottom of the panel, as shown in **Figure 2.5: Editor panel's standard buttons.**

-  **: Edit Default parameter values for this panel**. Allows for that specific panel to enter user-specified defaults that override the defaults in the program. A tooltip is provided, too. There is a button to restore defaults on the device. See **Figure 2.6: Edit Defaults dialog for Models > Soot Model > Settings .**
- **Defaults**: Set all of the parameters to default values specified by the user.
- **Apply**: The panel values are applied to the inputs. Changes don't take effect until clicking on the Apply button.
- **Reset**: The values are reset to the last "applied" configurations.

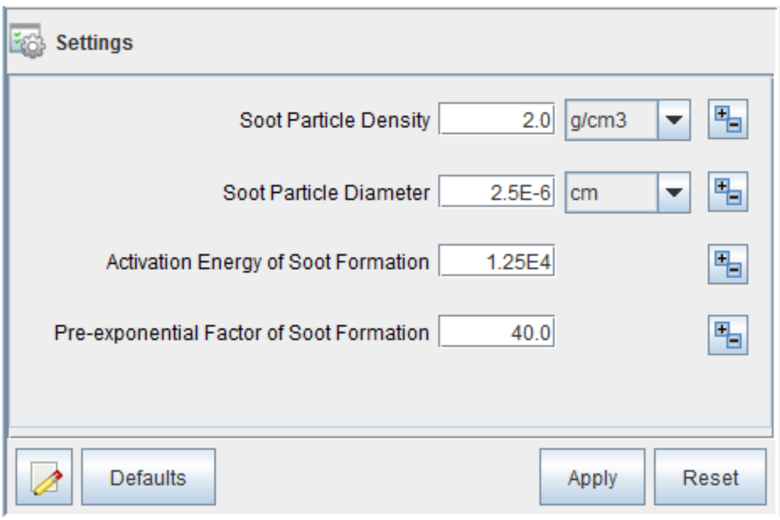

FIGURE 2.5: EDITOR PANEL'S STANDARD BUTTONS

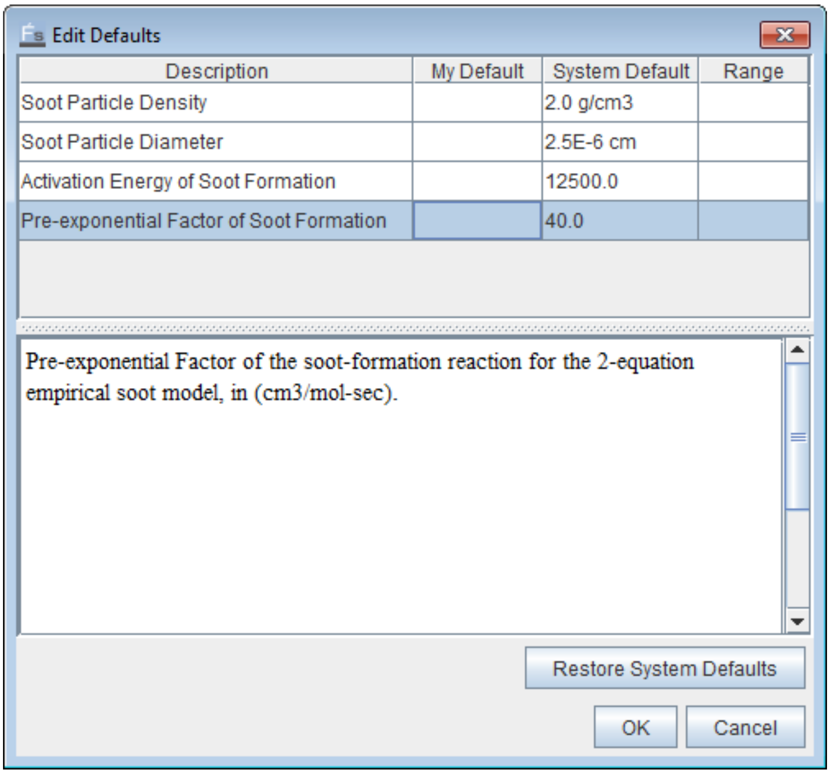

FIGURE 2.6: EDIT DEFAULTS DIALOG FOR MODELS > SOOT MODEL > SETTINGS

2.3.2. Expand/Collapse Button (+)

The expand/collapse  symbol enables sub-panel extension or collapse of several editor panels. The condition of the sub-panel setup can be changed in User Preferences.

2.3.3. Entering Profile Data

Many configurations of the model can allow the use of tabular profile data. ANSYS Forte provides the editor for the development, alteration, import and visualization of the profile data. See Profile Editor for information on working with the Profile Editor.

2.5. 3-D View Panel

Everything that can be visualize as: geometry surfaces, computational mesh, and/or boundary definitions, as well a spark and spray location information, are shown in the 3-D View panel bath the center of the simulation window. In the following points it is described how to modify the view using the mouse and the possibilities of the Visibility tree, each action has an effect on the 3-D view panel.

2.5.1. Mouse Behavior

The mouse allows to rotate or zoom the model in the 3-D View panel,. The default behavior of standard 3-button mouse is described in **Table 2.7: Default mouse modes**.

Table 2.7: Default mouse modes

Desired Effect	**Mouse Action..**
Rotate model	Left-click and drag
Zoom in	Middle-click and drag to right, or use scroll wheel
Zoom out	Middle-click and drag to left, or use scroll wheel
Pan	Right-click and drag in desired direction

2.5.2. Setting Mesh Display Attributes

The visibility tree manages each object's display properties, visibility and current color in the 3-D View screen, or right-click in the Workflow tree. To move an object between the visible and the unseen in the 3-D View, turn it on / off by pressing the light bulb in the Visibility tree for that object. Select Color in the context menu to adjust the color of an object, then pick a new color from the Color Selector.

The right-clicking in the Workflow tree or Visibility tree also offers controls for the following:

- **Opacity**: A range of opacity to be selected for the desired items in the Visibility tree between Opaque, High, Medium, Low.
- **Instancing**: Opens a dialog to assign criteria for different instances of the products selected. The number of new instances can be defined, and their location can be

interpreted and rotated relative to the original. Remember that this is for accessibility reasons only; it does not establish any additional underlying structures.

- **Order**: Requires to determine the relative order of drawing of selected objects to verify which objects appear at the end. To change the stacking order, pick Bring forward, or Send backward.

2.6. User Preferences

User preferences are the overview of many interface properties where configurable. To start the Preferences Editor dialog select Edit > Edit Preferences menu item in the toolbar. The expectations are classified into 4 groups, Display Settings, Preferred (3rd Party) Applications, Units Preferences, and File Preferences. A summary screen display of the panel Units Priorities is shown in **Figure 2.6: Edit Preferences dialog.**

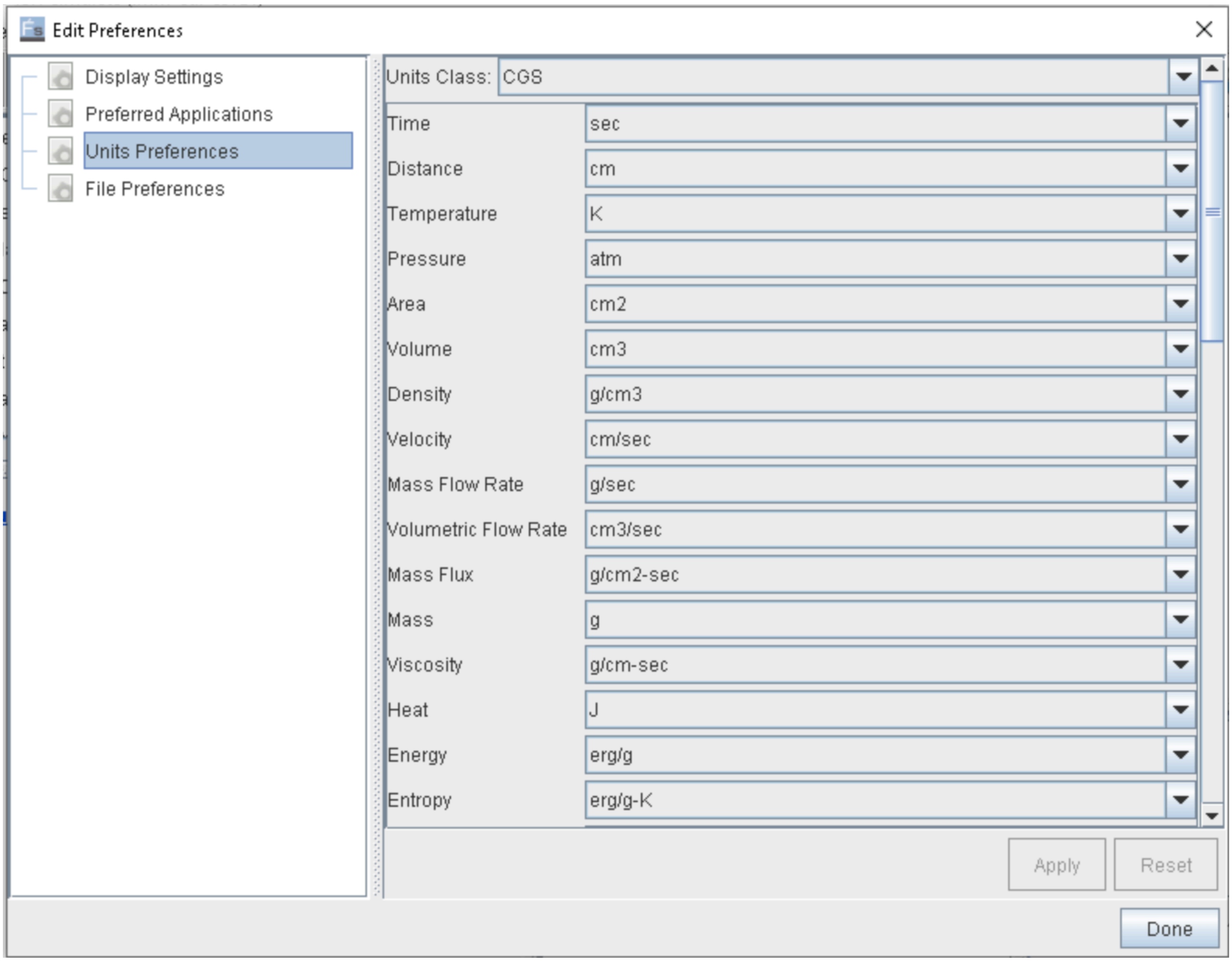

FIGURE 2.6: EDIT PREFERENCES DIALOG.

2.6.1. Display Settings Preferences

Items under Display Settings Preferences regulate the display window's visibility and attributes of some features such as the X-Y-Z orientation marker and background color. If the right-side Visibility tree is to be turned ON (the default is OFF), control its visibility here. The Visibility tree enables properties of collection and de-selection and accessibility visibility of objects in 3-D view.

The Show Information Labels preference setting controls the visibility of labels. Turning it OFF removes point marked view in a point cloud.

2.6.2. Preferred Applications Preferences

In the Preferred Application Preferences panel, it is possible to decide which 3rd-party program to launch while opening other types of files from the interface of ANSYS Forte. It is recommended using the default associated with the operating system of the computer and only change this setting if this option is not working for some reason or you want some features of an alternative application.

2.6.3. Units Preferences

The Units Preferences panel allows control of the default units for various types of physical data, as entered in the Simulation Interface Editor panels. See in **Figure 2.7.**

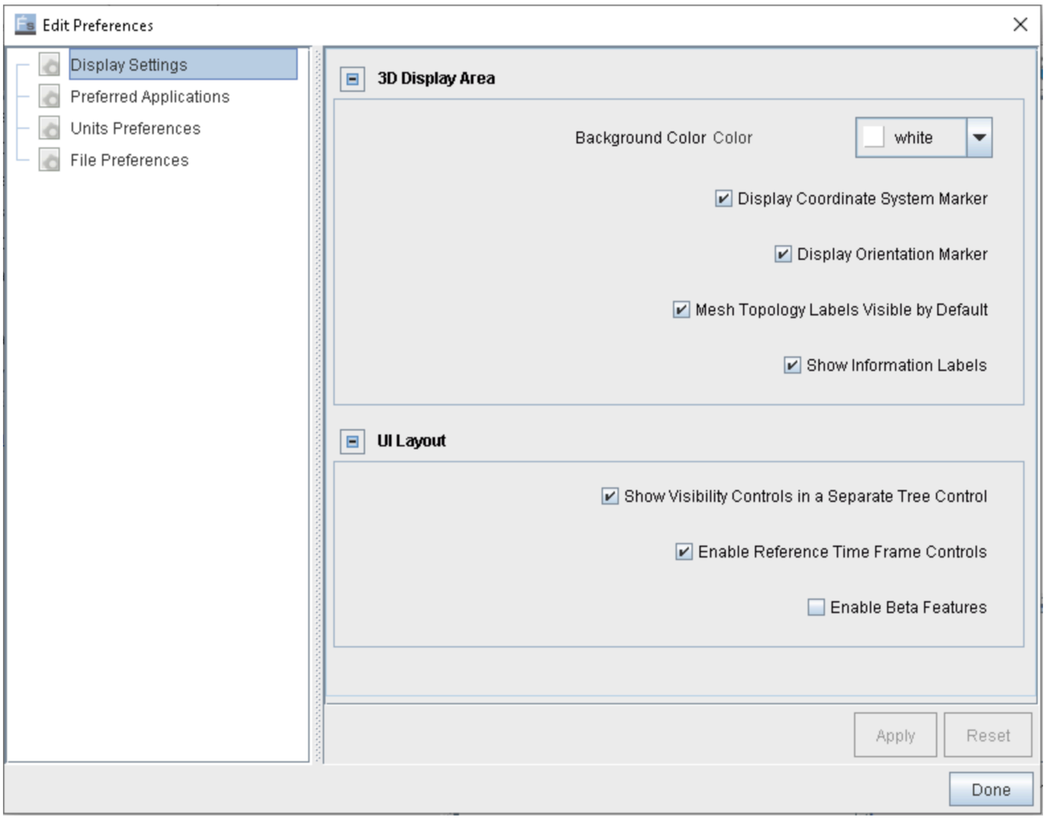

FIGURE 2.7: DISPLAY SETTINGS DIALOG.

2.6.4. File Preferences

In the **File Preferences** panel, Allows users to monitor the default working directory, which specifies where the file explorer begins when the program is first opened for various file searching tasks, as well as file search history. If these are not allowed to be included in the options list during file selection actions, it can be deleted history entries. See in **Figure 2.8: File Preferences.**

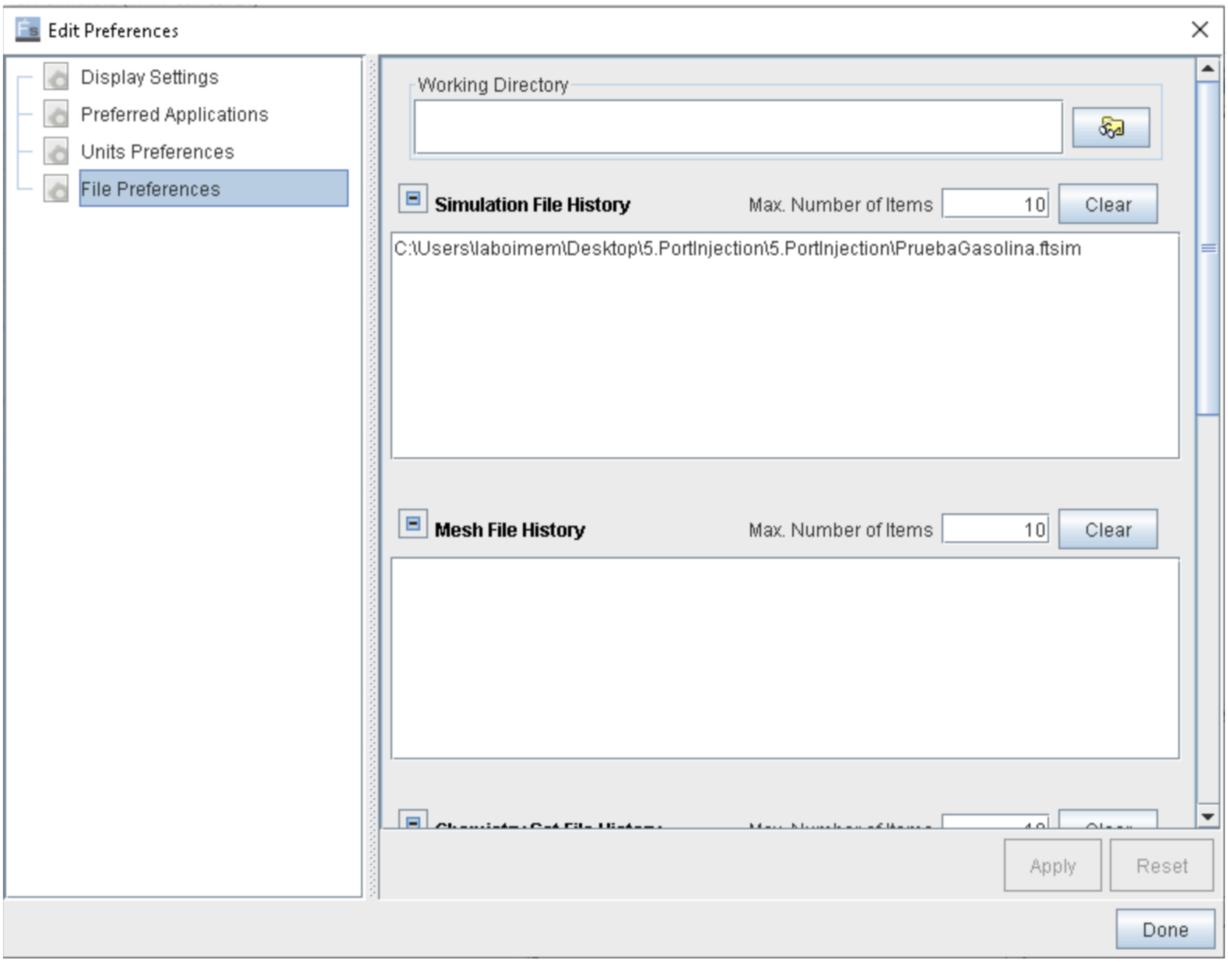

FIGURE 2.8: FILE PREFERENCES.

2.7. Data Entry and Management Tools

ANSYS Forte offers multiple data entry resources needed for simulations, and many of the options are data import, upload, and handle. The following subsections identify such devices.

2.7.1. Profile Editor

In the Editor panel is used the **Edit Profile**  button to introduce data whenever is required. This option will display the Profile Editor shown in: **Figure 2.10: Profile Editor with data imported from a CSV file.** The Profile Editor allows to manually enter data or paste from Microsoft ® Excel ® or a similar program.

In the **2D Profile** list provides sine and square profiles ready to use, see in **Figure 2.9: Profile Editor selection dialog.**

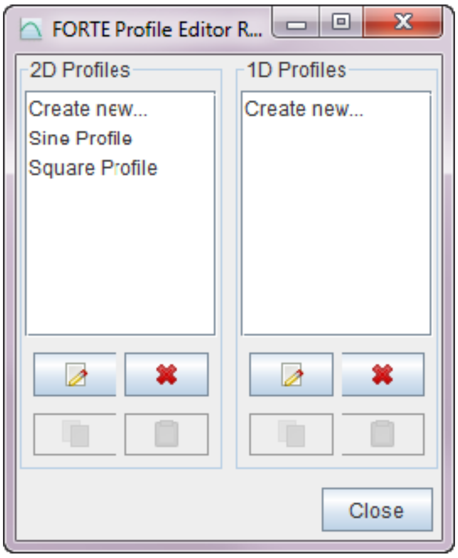

FIGURE 2.9: PROFILE EDITOR SELECTION DIALOG

When an external table has been introduced, the units for the columns of there profile must have been specified. Instead of the command "copy and paste" there is an option to introduce tables in a comma-separated values (CSV) text file.

The Profile Editor panel plot the profile once the data is entered in the columns as shown in **Figure 2.10: Profile Editor with data imported from a CSV file**. There are options in the lower down panel to change the name and save the file.This will return to the Editor panel from which the Profile Editor was launched.

Verify that the latest edited or generated profile is now selected in the Editor panel for the desired choice. Sometimes there is a mistake and the program does not select the new one automatically.

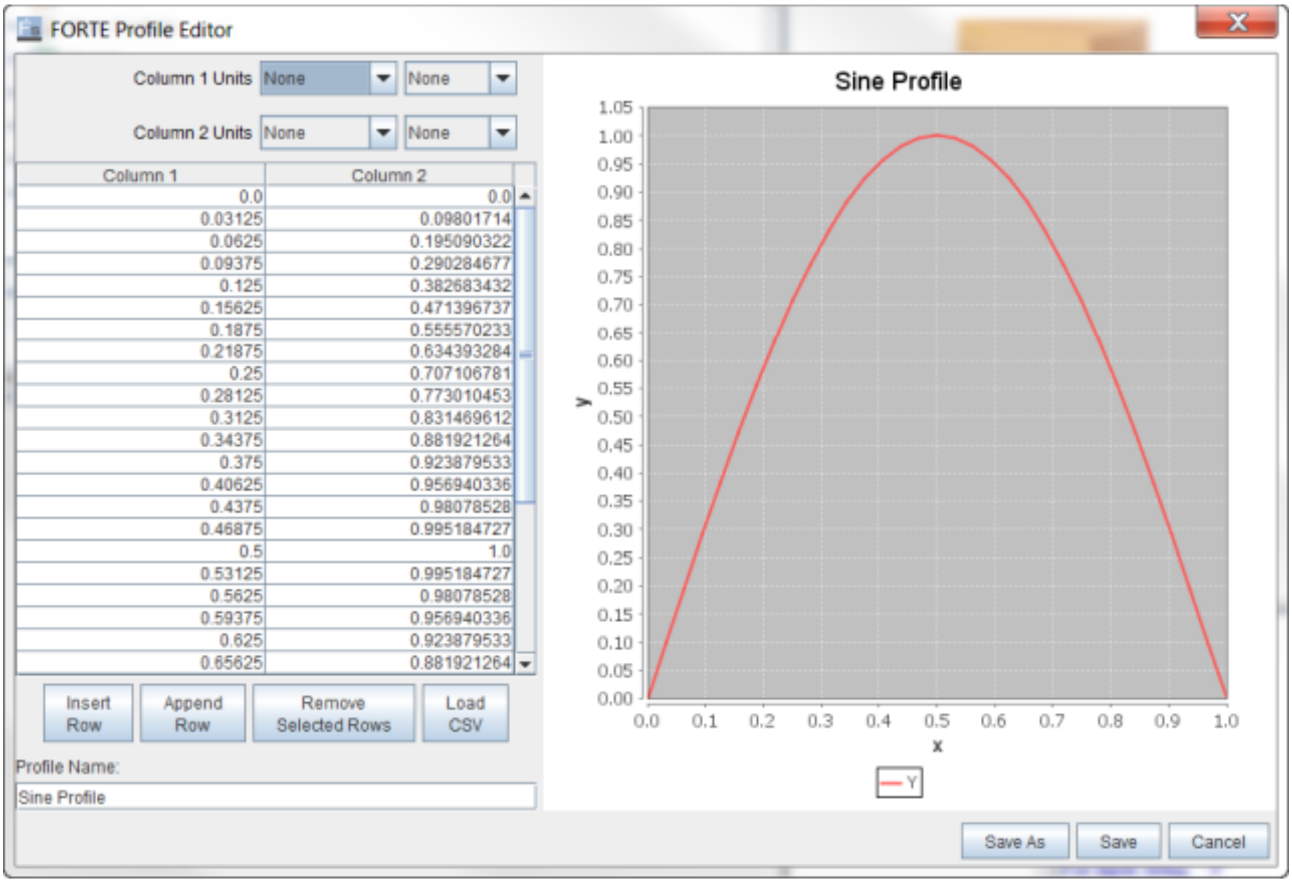

FIGURE 2.10: PROFILE EDITOR WITH DATA IMPORTED FROM A CSV FILE

2.7.2. Composition Calculator

The Composition Calculator is used to create liquid or gaseous fuel, is able to introduce any $C_xH_yO_zN_w$ composition, for rich or lean conditions, with and without EGR (internal and/or external EGR).

For the calculations complete composition is assumed. The composition to be measured contains:

1. Intake port/inlet composition for engine cases with valves and ports.
2. Exhaust port initial composition for engines cases with valves and ports.
3. In-cylinder initial composition at IVC for engine cases that are in-cylinder only.

The **Composition Calculator**  button displays the panel where the data should be introduce to create the mixture. The options fuel mass, fuel composition, air flow data, and EGR data must be full field for the calculations, the results will be displayed in a table format, as illustrated in **Figure 2.11: Composition Calculator utility**.

Once the mixture has been calculated click the **Create Mixture** button to save the results and use them when such composition input is required. Click **Save** button to save current IVC session. Using the Open button to load a saved session as a ***.ftivc*** file into the calculator

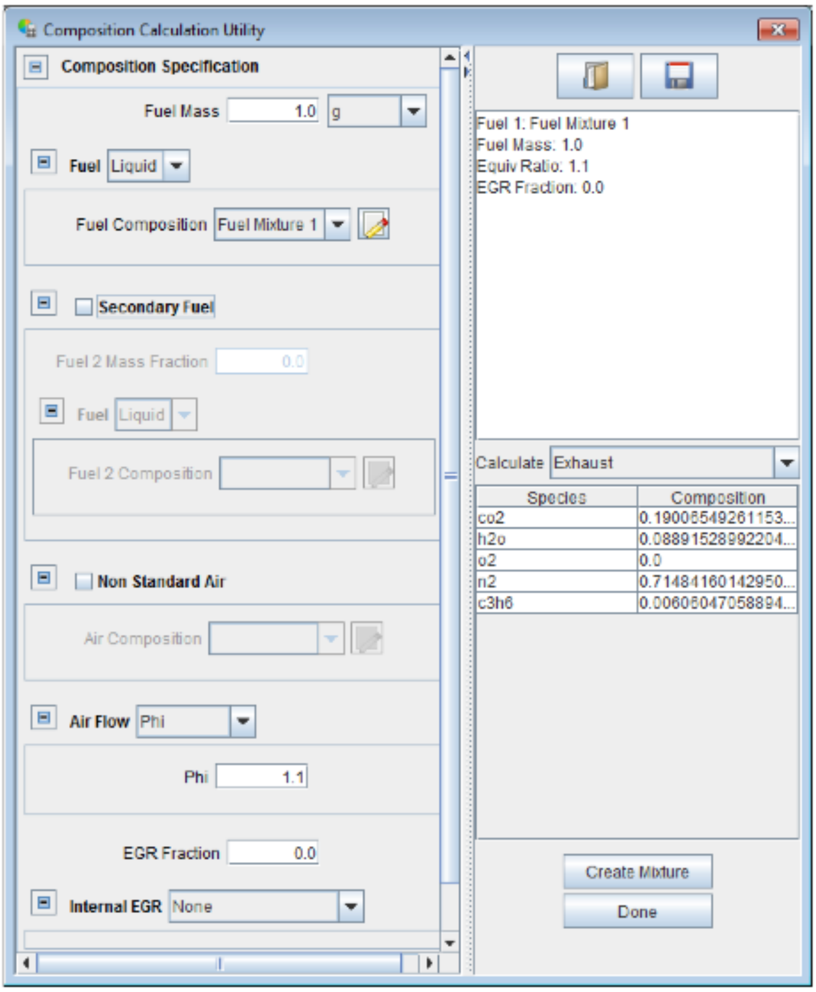

FIGURE 2.11: COMPOSITION CALCULATOR UTILITY

2.7.3. Mixture Editor

Mixture Editor button can be accessed from the Edit menu, the button on the toolbar or from a panel, such as Initialization, with a Composition parameter.

This Editor is used to specify the species for a composition needed. There are two options for the measure of the species: **Mass Fraction** or **Mole Fraction**, then use the **Add Species** button to select species that are present in the gas mixture. In the resulting table, specify the associated fractions for each species. See in **Figure 2.12: Creation of a gas mixture.**

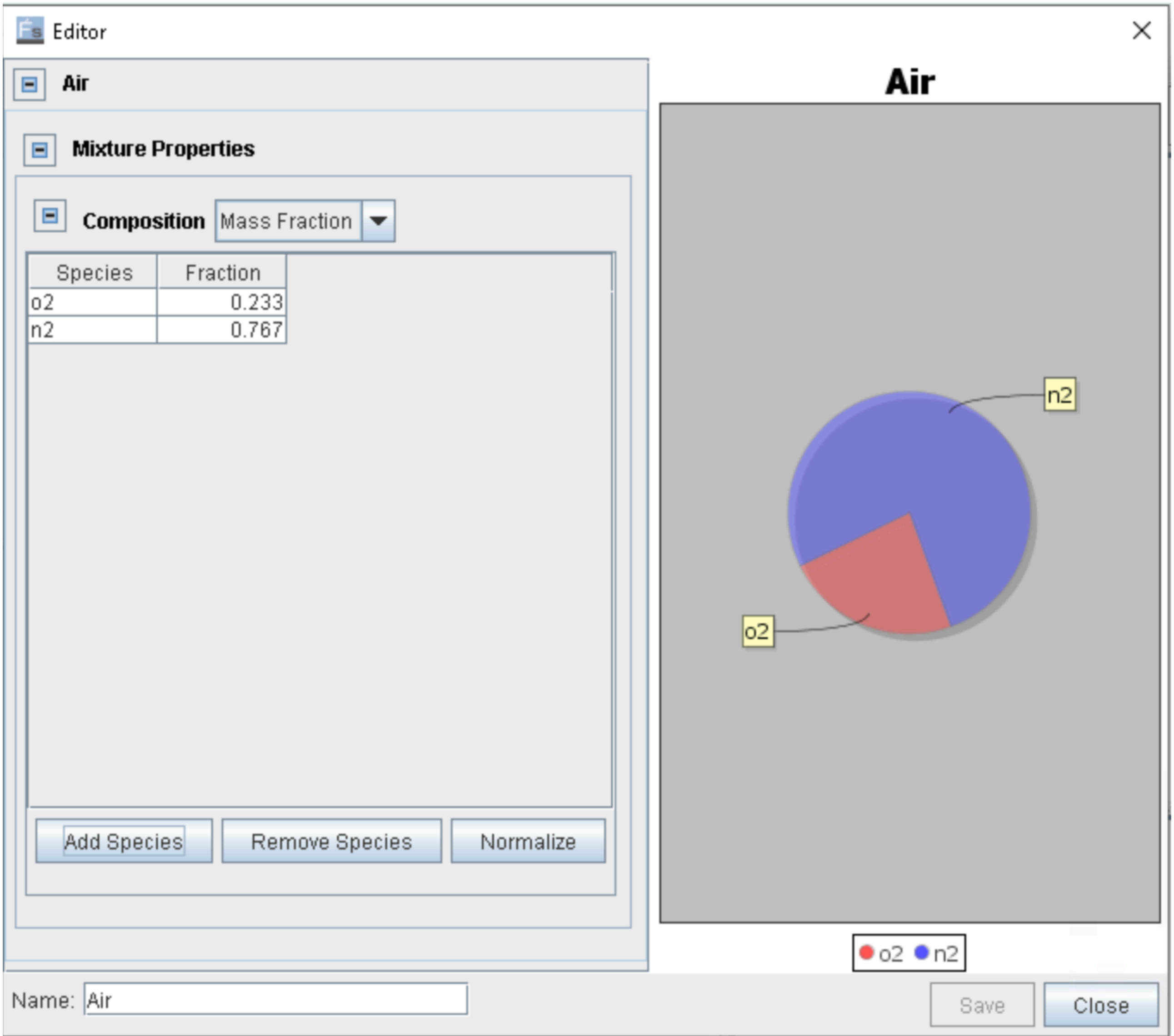

FIGURE 2.12: CREATION OF A GAS MIXTURE.

2.7.4. Initial Conditions Table Editor

From the **Edit** menu or from the Initial Conditions panel or the button on the toolbar the Initial Condition Table Editor can be accessed, where the spatially resolved initial conditions are defined.

Initial Tables of Condition take the format of a ***.ftbl*** file, a .***cgns*** file or a standardized collection of ***.csv*** data.

The Initial Conditions Table Editor shows the required data collection and offers the possibility of importing file source replacement data.

2.7.5. Boundary Condition Table Editor

Spatially varying boundary conditions are specified in he Boundary Condition Editor. The Boundary Condition Table Editor can be reached from the Boundary Conditions panel or from the **Edit** menu or the button on the toolbar.

2.7.6. Flame-Speed Table Editor

To use the Flame-Speed Table Editor, click on the Edit menu or the **Flame-Speed Table** button in the toolbar, where you can build or update a list.

A flame speed look-up table method is used in ANSYS Forte to determine laminar flame speeds, as feedback to the flame-propagation model. The look-up table is a file that includes a set of laminar flame-speed data points used during the simulation to interpolate to a suitable, local flame level

2.7.8. Parameter Studies

ANSYS Forte allows to modify the simulation parameters and models by using the **Parameter Study** **.** This ability is an importan feature to design parameter studies in which simulation parameter can vary over a series of runs.

2.7.9. Reference Frames

There is an option for creating reference frames within the Geometry node. This reference frame, once defined, can be used to define points, directions, and velocities in the different Simulate Editor panels. Cartesian, circular, or cylindrical coordinate systems can be used to identify or compare frames of reference.

To create a new frame is to go to Geometry > Reference Frames on the Workflow tree and click the **New Reference Frame** button. Select a name for the reference frame, and specify the properties, including **Origin**, **Coordinate System** and coordinate values, and optional rotation. See in, **Figure 2.12: Reference Frame creator**. Once is created it is added to the items in the workflow tree.

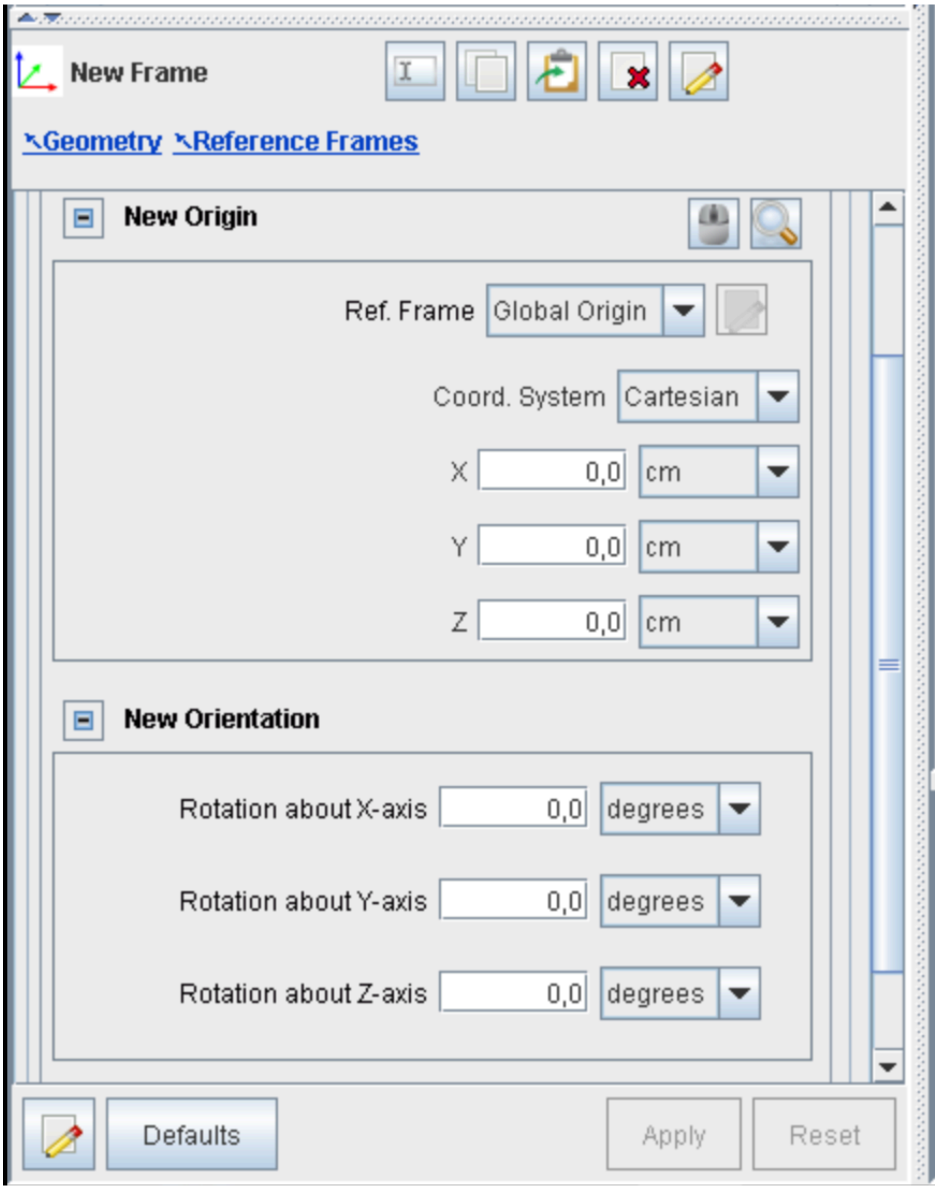

FIGURE 2.12: REFERENCE FRAME CREATOR.

2.7.10. Time Frames

An choice to determine the Time Frame exists in several Editor panels. It allows for setting a time frame of reference. This time-frame offset, if established, can then be used in controls of precision, sprays, sparks, and movement. Standard time-frame checks are covered.

2.7.11. Clip Planes

The **New Clip Plane** button shows the panel of the clip editor, which is used to determine the location of the clip plane. This device produces a cutting plane where the actually observable surfaces are separated. The monitor eliminates every surface that is in front of a clip plane. This is helpful when showing features within a watertight surface that is locked. It is formed just like a new frame.

2.7.12. Sub-Volumes

Sub-volumes are valuable for refining of interest regions, such as the chamber, see in **Figure 2.13: Chamber selection**. The sub-volume will then be used to monitor the scale of the mesh in the chosen sub-volume inside the mesh controls.

New Sub-Volume : Generates a sub-volume to define a standardized mesh size or a mesh size greater than the global mesh size. They can also be used as the basis on which to define initial conditions.

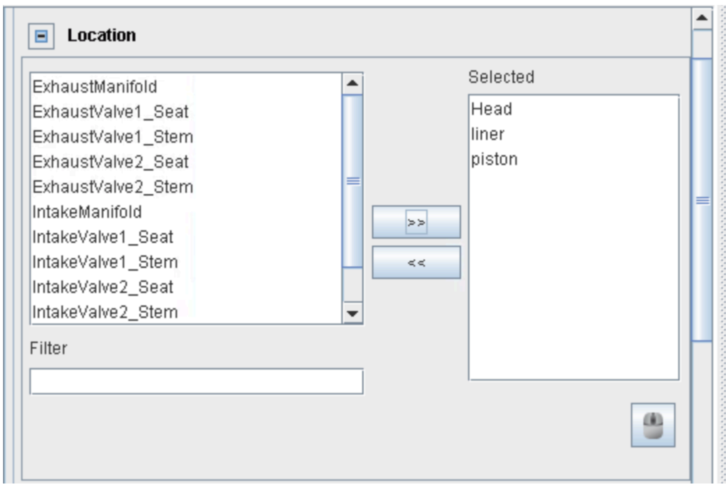

FIGURE 2.13: CHAMBER SELECTION

2.7.13. Compression Ratio Calculator

The Compression Ratio Calculator can help decide how to change a piston's position to reach a specified compression ratio, which can be used to validate a specific configuration. This usefulness is true for a chamber which has a crank-angle-based motion control applied to the position of the moving boundary. Link to the compression ratio converter is available in the Utility screen.

The utility uses the measurement method No-Hydro CR which runs a single time step at two α and β time points; these are the points where the angles of the cranks are at TDC and BDC. This approach uses the generation of the solver mesh to achieve an exact estimate of the volume of the in-cylinder at any point in time.

2.7.14. Solid Phase Editor and Dispersed Phase Editor for Particle Tracking

The Solid Phase Editor can be accessed from the **Edit** menu or the **Solid Phase Editor** button on the toolbar, or from the Initial Conditions or Inlet editor panels.

The Solid Phase Editor is used to add additional particles such as the present original particles, or the particles in an inlet.

Specifying the fraction of the solid-phase surface uses an editor similar to the one described in Mixture Editor for gaseous composition. Add the species, and their fraction, the editor is show in **Figure 2.12.**

3. Modeling Guide

The modeling nodes for the Simulation Workflow tree are explained in this point. ANSYS Forte has been designed for precise mechanics, spray physics and chemistry simulation of internal combustion engines.

3.1. Geometry Node

For the Simulation Workflow tree the Geometry node is the beginning. This offers options to load predefined mesh or geometry files, generate sector mesh and create conditions for automated mesh generation during simulation.

Under the node Geometry the surfaces are identified. The Editor panel below the Workflow tree provides details about the surface's limiting coordinate values and certain behavior that can be added such as: **Rename**, **Split Mesh**, **Transform Mesh**, and **Invert Normals**.

In the following subsection it is explained the icons related with mesh creation, including **Launch Sector Mesh Generator**, **Import Geometry**, **Export Geometry**, and **Merge Meshes**, **Join Meshes**, **Measure Geometry**, and **Check Surface Mesh**.

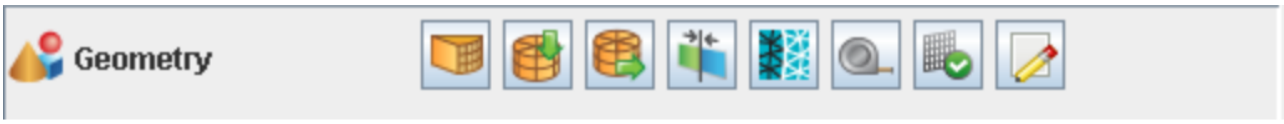

FIGURE 3.1: GEOMETRY COMMANDS.

3.1.1. Sector Mesh Generator

The **Sector Mesh Generator**  icon launches a window where a body-fitted sector mesh is created using a minimum number of user specifications which describe the piston movement and bowl shape of the engine.

Similar to ANSYS Forte Simulate, the Sector Mesh Generator window has nodes for Engine Parameters and Mesh Parameters, while the mesh and mesh function settings are shown in the 3-D View window.

The Engine Parameters panel can define values for the profile of the piston bowl, piston-motion characteristics, sector angle or sector cyclic symmetry, as well as the dimensions of the engine-cylinder and the size of the crevice.

The Mesh Parameters panel controls the mesh density and topology as shown in **Figure 3.2: Sector Mesh Generator interface**.

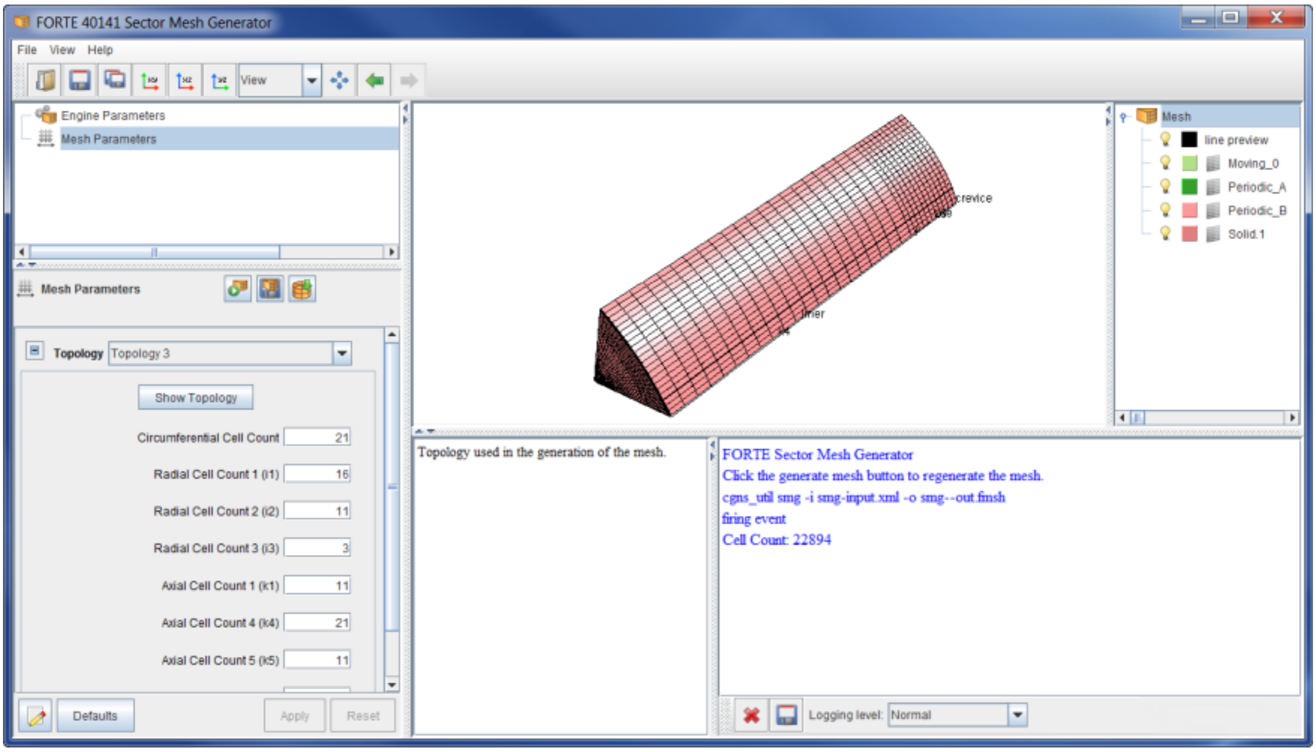

FIGURE 3.2: SECTOR MESH GENERATOR INTERFACE

3.1.2. Import Geometry

The Geometry panel gives the option to import geometric surfaces from various forms files, with the **Import Geometry**  icon, as well as pre-defined body-fitted meshes.

Whenever a geometry file is imported, it must be specify the information of the type of file. The following files are accepted:

1. Body-fitted Mesh from the ANSYS Forte project.
2. Body-fitted Mesh from KIVA-3V (itape17) Format (ANSYS Forte ***.fmsh***).
3. Surfaces from CGNS exported from another CFD program.
4. From FLUENT mesh file.
5. Surfaces from STL file.

There is a choice in option 1and 2 between importing the whole mesh or only the geometry and using the simulation automated mesh creation. The length of the STL data must be specified when selecting an STL file to be imported.

3.1.3. Export Geometry

In the Geometry panel, the **Export Geometry**  icon displays a widow to select the location of the file and save it with a name as an STL file defining all the surfaces actually found in the Geometry node See in, **Figure 3.3: Export utility's Save dialog**

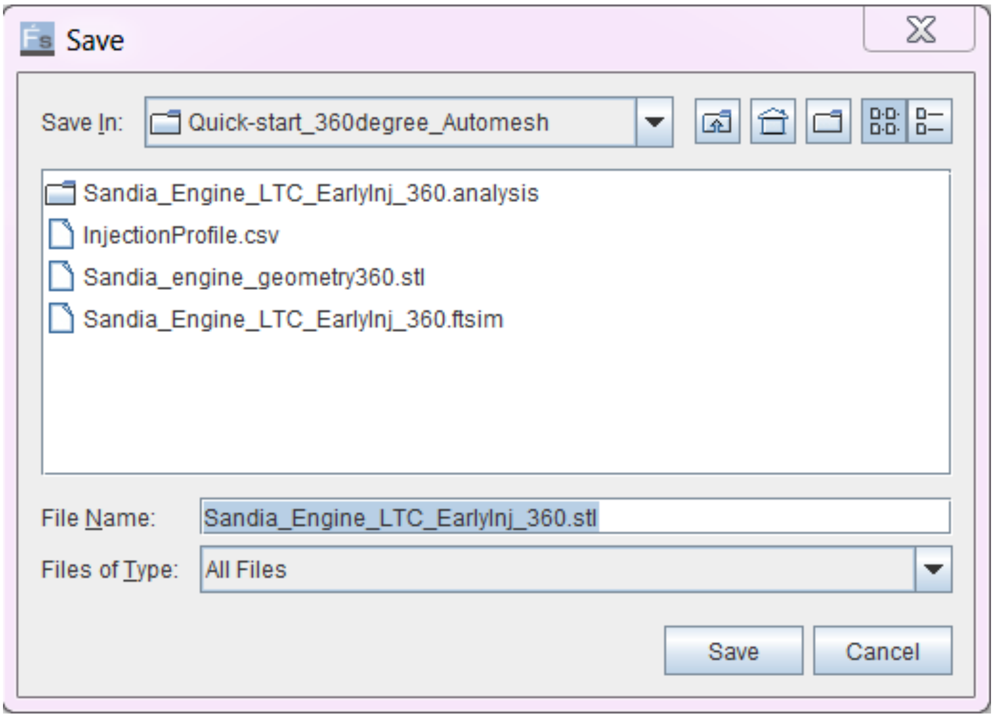

FIGURE 3.3: EXPORT UTILITY'S SAVE DIALOG

3.1.4. Merge Meshes Utility

The **Merge Meshes**  icon on the Geometry panel allows multiple mesh elements to be fused into a single geometry surface, it can only be used with the option of automatic mesh generation. The Duplicate edge tolerance choice determines the sensitivity of the chosen surfaces to merge the edges between them. See in **Figure 3.4: Surfaces to merge.**

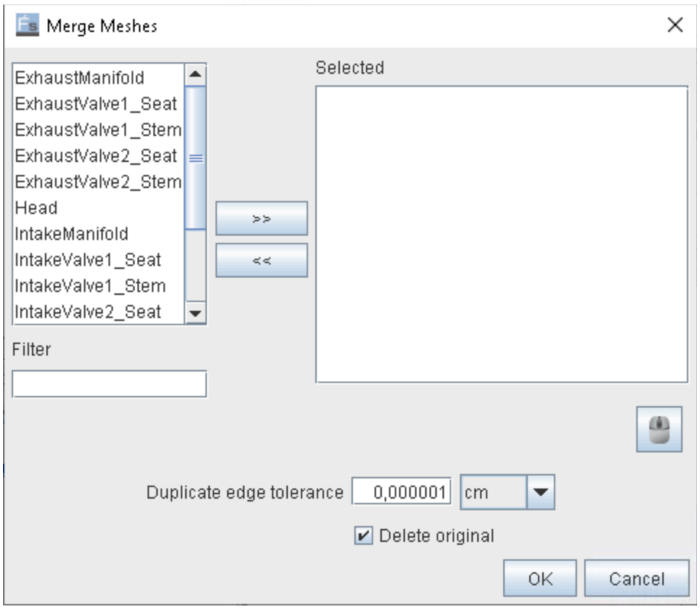

FIGURE 3.4: SURFACES TO MERGE.

3.1.5. Join Meshes Utility

The **Join Meshes**  icon on the Geometry panel helps to patch the open edges. This will happen when a surface mesh with a low resolution has been imported. Red highlights on surface edges will define this leaky mesh problem. This menu is similar to the Merge Meshes shown in **Figure 3.4: Surfaces to merge.**

3.1.6. Measure Geometry Utility

To measure lengths, angles, and other properties of geometric structures, using the Measure Geometry Utility .

This button opens a dialog with options for setting 2- or 3-point metrics that allow you to choose from the Workflow tree using reference frames, or by choosing an existing position. Shown in **Figure 3.5: Measure menu.**

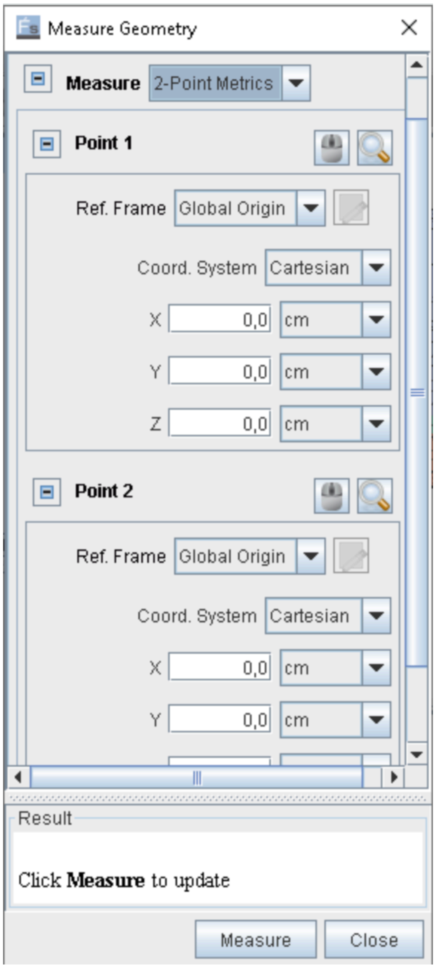

FIGURE 3.5: MEASURE MENU.

3.1.7. Surface Integrity Check Utility

In case of the surface has problems that could lead to failures while running the simulation, there is a tool  which checks the surface mesh geometry

If there is a problem it will appear in the Geometry node as a new item called **Problem Vertices.”**

By default, when a run is submitted for execution, the Surface Integrity Check utility runs automatically, and will terminate the run if problems are found. This feature can be disabled in the Workflow tree's Run Settings element in the case of a clean surface that causes problems.

3.2 Mesh Controls for Automatic Meshing

The Mesh Controls node displays different options due to the import geometry, in this guide it is explained the Mesh Controls for the use of automatic mesh generation.

The Mesh Controls workflow tree is split in the following sub-nodes: Material Point and Global Mesh Size. See in **Figure 3.6: Mesh Controls sub-nodes.**

FIGURE 3.6: MESH CONTROLS SUB-NODES.

- **Material Point**: The Material Point must be located inside the cylinder, in a position that is always inside the geometry during the entire cycle and is at least one unit cell length away from any boundaries.
- **Global Mesh Size**: Affects meshing and refinement throughout the system.
  - **Mesh Size**: Specifies cells size, usually as a fraction. The solution time is related with the length choose value. Units are user-selectable.
  - **Small Feature Deactivation Factor**: It is used to select the treatment for small-scale surface features smaller than a cell size, as sharp corners. Range = 0 to 1.

3.2.1 Refinement Mesh Types

From the Mesh Controls node can be added different refinement types, depending on the geometric features. it must be specify in each mesh the name, location, size fraction, cell layers and the activation, it could be always or in a specific time step. See in **Figure 3.7: Refinement Mesh Types**, the different types are describes below:

FIGURE 3.7: REFINEMENT MESH TYPES

- **Point Refinement Depth**: A sphere of mesh-size is generated, where the parameter needed are location and radius.

- **Surface Refinement Depth**: This tool allows to mesh any surface defined in the geometry.

- **Feature Refinement Depth**: It is applied along edges whose feature angle is greater than a specified feature- angle threshold value.

- **Line Refinement Depth**: It allows to specify the mesh-size for any line along the geometry.

- **Small Feature Avoidance Control**: Creates a feature with the Small Feature Deactivation Factor applied. Define a Location, Small Feature Deactivation Factor.

- **Secondary Volume**: Creates a mesh-size for the sub-volume selected.

- **Solution Adaptive Meshing**: Adaptive meshing solution allows the mesh to be refined at the current time step, based on a solution field (or gradient). Both the selection criteria and the area of solution must be defined. There are three types of criteria:: Statistical, Percentage of ocurrente solution bounds, Absolute value.

3.3. Models Node

The Models node allows specific models to be implemented and options to be identified relating to those models. Using the check box, if a pattern is chosen, more choices for configurations will appear. See in **Figure 3.8: Models options.** The distribution of the models features.

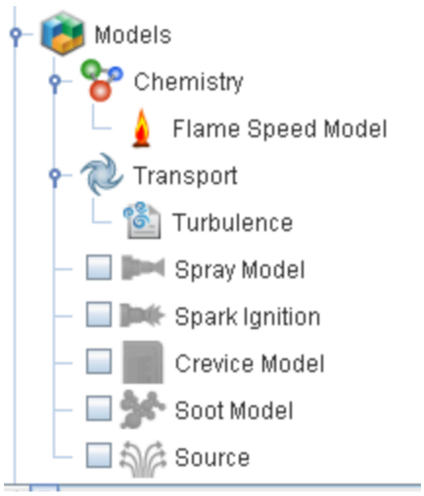

FIGURE 3.8: MODELS OPTIONS.

3.3.1. Chemistry

In the Chemistry panel there is an access to the **Import Chemistry** icon on the Chemistry icon bar. Browser window displays to select and load a chemical reaction mechanism from a pre-defined Chemkin-Pro chemistry set file

View Chemistry is the second icon on the icon bar, which opens a panel showing information relevant to the chemistry package being imported.

The third icon is the **Reload Chemistry Set**, which opens the chemistry-set information to a file browser.

There is an alternative to create a chemistry file in the **Utility** menu. The chemistry collection decides the chemical species or molecules in the simulation are possible

3.3.1.1. Flame Model

ANSYS Forte lets it assign multiple flame speed choices. The laminar flame intensity may either be measured using power law formulas, or using pre-calculated flame-speed tables.

3.3.2. Transport

There are two sections in the Transporte node; Turbulence and Fluid Properties. In the Editor Panel are represented the parameter for the calculation of the turbulent fluid properties. It is recommend accepting the defaults under Fluid Properties for most engine cases.

3.3.2.1. Turbulence

The turbulence Editor panel allows two options for the calculus: The **Turbulence Model**: **RANS** (Reynolds-Averaged-Navier-Stokes) or **LES** (Large-Eddy_Simulation) models.

- For the **RANS** models, options of **RNG k-epsilon** or **k-epsilon** are used.
- For **LES** models, the options of **Smagorinsky** or **Dynamic Structure** model are offered.

- For laminar flow simulation **no model** is selected.

ANSYS advises that the RNG-k-epsilon configuration be used in engine cases and in spray chamber cases. Also, the RNG-k-epsilon model would be a better engine combustion model

3.3.3. Spray Model

Whenever a Spare injection is needed, it must be check the box to enable it (uncheck by default).

Inside this node can be configured: Droplet Collision Model, Injectors, Nozzles, and Injections.

For the Droplet Collision Model option The following options are available:

- Disabled, Radius of Influence Model
- Collision Mesh Model,
- Adaptive Collision Mesh Model and specify the associated options.

Using Vaporization Model indicates vaporization (default) of the injected liquid is permitted. This choice is needed for liquid-fueled motor cases.

When injector is created, the composition of the fuel model must be specify and also the geometry for the ejector full filling the parameter for nozzle geometry and injection events, Solid-Cone, Hollow-Cone, and Slit Injector Panels, as well as the spray-model settings. See in **Figure 3.9: Geometry spray icons.** The buttons for Solid-Cone, Hollow-Cone, and Slit Injector Panels.

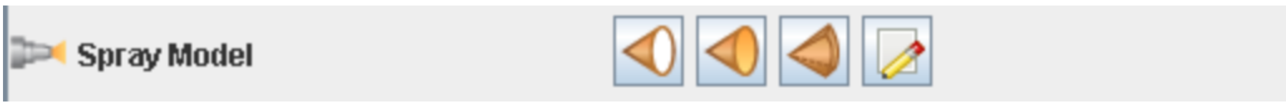

FIGURE 3.9: GEOMETRY SPRAY ICONS.

3.3.3.1. Solid-Cone, Hollow-Cone, and Slit Injector Panels

The Injector icon bar offers the following icons that are used to define the nozzle holes and injection events associated with the injector:

1. **Rename** : Chosen nozzle name.

2. **Manage Injections** : Provides worksheet to define properties for the injection.

3. **New Nozzle** : Adds an injector-associated nozzle and opens the Nozzle panel to customize the new nozzle.

4. **New Injection** : Adds an injection case connected with the injector (and nozzles) and opens the Injection panel for timing and fuel quantity adjustment of the injection.

5. **Copy** : Copies the injector along with any injections or nozzles connected to the injector.

6. **Paste** : The copied injector is pasted along with it is specified nozzles and injection events.

7. **Delete** : Clears the injector.

The settings of the spray-model are regulated in the central Editor panel of Solid-cone Injector.

- **Composition**: Configure the Fuel Composition.

- **Time Frame**: Set the time offset for all subsequent injections.

- **Injection Type**: Pick pulsed or a constant injection. Hollow injectors are used for pulsed injection only.

- **Parcel Specification**: Specify the number of parcels injected, either for pulsed injection or constant injection. Only Droplet Density can be defined for continuous injection in an injected parcel.

- **Inflow Droplet Temperature**: In an engine setting, the initial temperature of injected droplets is usually similar to the temperature of the coolant.

- **Droplet Size Distribution**: To trigger spray injection, determine the size distribution of droplets.

There are several settings choices for the Solid-cone Injector which are unique to this type of injector.

- **Spray Initialization**: The spray can be initialized using a Discharge Coefficient empirically derived, or using a more precise Nozzle-flow Model.
- **Mean Cone Angle** is required for the injector.
- **KH Model Constants**: The Kelvin-Helmholtz near-nozzle droplet-breakup model uses certain parameters.
- **RT Model Constants**: These parameters are used in the droplet-breakup model of Rayleigh-Taylor (far-from-nozzle).
- **Use Gas Jet Model**: This alternative helps reduce the reliance on mesh size and time-step. Set is ON.

The Hollow-Cone Injector and Slit Injector both have multiple choices for setting the spray actions unique for hollow-cone injections:

- **LISA Spray Model Options**: Parameters for the LISA spray model are provided. During processes of spray forming the Breakup Period Model Constant determines the breakup duration of the liquid layer
- **Injection Pressure**: This should be the pressure of injection in the injector's sac volume. It is used for measurement of the coefficient of discharge.
- **Mean Cone Angle**: Just Sprays for Hollow-Cone. The parameter regulates the hollow-cone spray shape opening angle. **Figure 3.10: Definition of mean cone angle and liquid jet thickness in a hollow-cone spray.**
- **Liquid Jet Thickness**: Just Sprays for Hollow-Cone. That prescribes the spray layer spreading angle. **Figure 3.10: Mean cone angle and liquid jet thickness in a hollow-cone spray.**
- **Model Slug Flow**: To compensate for the slug flow (the residual fuel flow left in the injector from the previous injection), choose this option. The slug discharge is represented as a tiny stream of solid-cones. Usually it is a very small part of overall fuel flow and is not typically based on.
- **Slit Angle**: Only for fan sprays. It is the Slit Injector nozzle opening angle, which determines the spray's main fan angle as well. See **Figure 3.11: Definition of several injection and nozzle parameters for slit injectors (Left: front view; Right: side view)** (Left).

- **Side Spreading Angle**: Only for fan sprays. This function defines the angle of dispersion of sprays relative to the fan axis. See **Figure 3.11: Definition of several injection and nozzle parameters for slit injectors (Left: front view; Right: side view)** (Right).

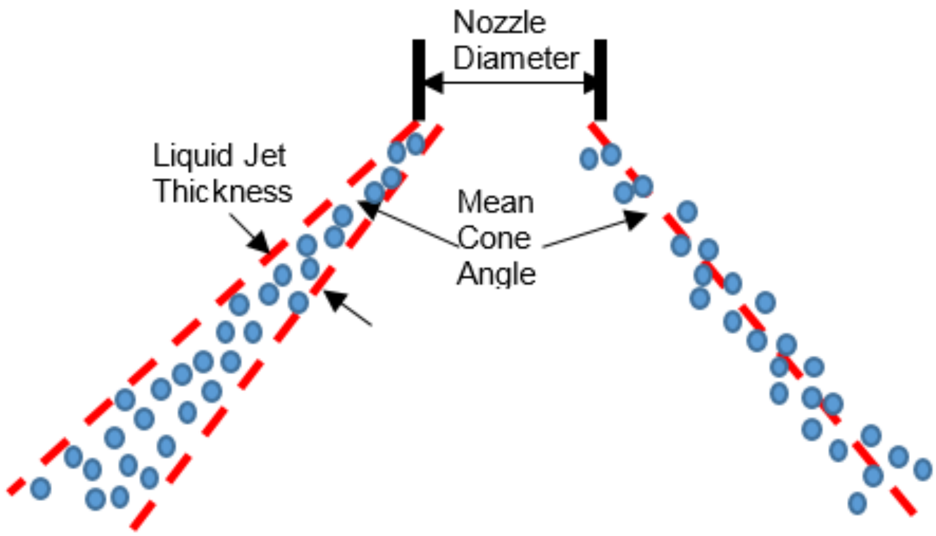

FIGURE 3.10: MEAN CONE ANGLE AND LIQUID JET THICKNESS IN A HOLLOW-CONE SPRAY

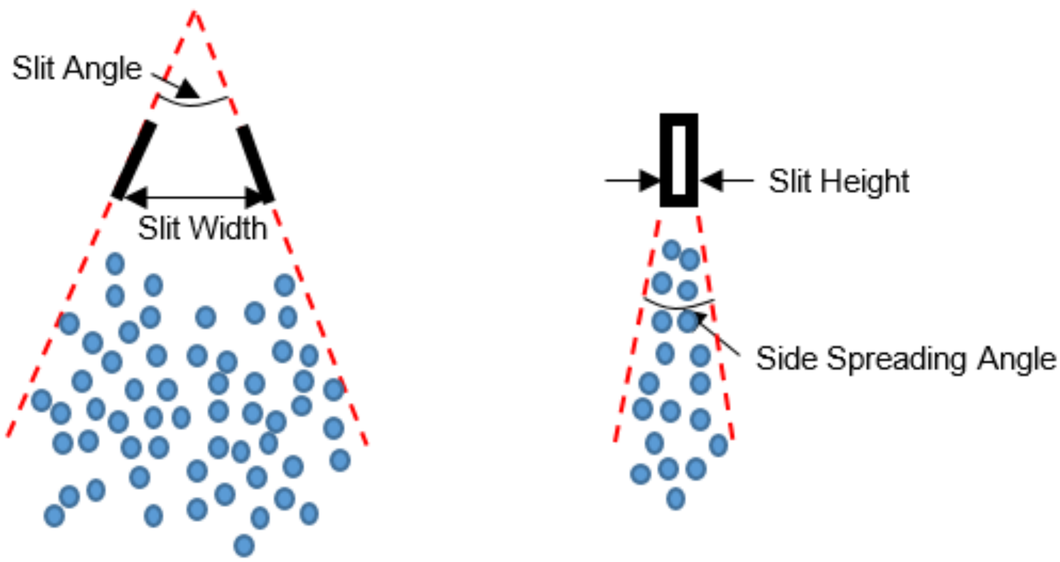

FIGURE 3.11: DEFINITION OF SEVERAL INJECTION AND NOZZLE PARAMETERS FOR SLIT INJECTORS (LEFT: FRONT VIEW; RIGHT: SIDE VIEW)

3.3.3.1.1. Injection Panel

The **New Injection** icon is used to specify he injection timing in terms of Time or Crank Angle. (using a Pulsed injection)

The Injection panel allows editing or making an Edit icon for the Velocity Profile and defining the total injected mass. The Velocity Profile option opens the Profile Editor (see Profile Editor) where it can manually define the profile or import data which defines the shape of the injection. For Continuous Injection, the Mass Flow Rate is only necessary.

3.3.3.1.2. Nozzle Panel

When a Nozzle is formed it must determine the position and geometry characteristics. There are also 4 icons allowed in the Nozzle icon bar: rename , copy , paste , and delete .

This helps create a new nozzle by copying and pasting an current nozzle and changing the parameter that needs to be modified. Using the sub-panel Guide to provide position and spray path of each nozzle hole

3.3.4. Spark Ignition

In the Spark Ignition Panel it can be configured the Spark ignition settings, as the number of sparks, and for each, the Spark Location, Timing, and Energy. There are also 4 icons allowed in the Spark icon bar: rename , copy , paste , and delete .

The Spark Ignition Editor panel provides settings that apply to all spark events in the simulation:

- **Kernel Flame to G-equation Switch Constant**: Enter a value for the transformation from configuration of the ignition kernel to that of the G-equation. This transformation occurs as the radius of the kernel becomes greater than this ratio, twice the integral length scale of the turbulence. Scale suggested: 1.5–2.5.

- **Min. Kernel Radius for Kernel to G-equation Switch**: Until they may turn to the G-equation model, the spark ignition flame kernel radius will expand greater than this minimum radius. Width suggested: 0.1 cm.

- **Flame Development Coefficient**: This coefficient governs the rising exponential effect of turbulence on the speed of flame propagation as the flame rises from laminar to fully formed turbulent flame. The transformation is speeded by a greater value. Scale suggested: 0.5–2.5.

- **Number of Flame Particles for each Spark Plug**: Amount of particles used to map a spark ignition kernel flame's flamma front position. On all spark plugs this value is applied. Value suggested: 3000.

3.3.4.1. Spark Panel

Once the Spark ignition is defined, the Spark must be define top. Using the **New Spark** icon on the Spark Ignition icon bar it allows to specify the parameter for the Spark. The Spark icon bar also offers 4 icons: Rename, Copy, Paste, and Delete, which are used to manage the sparks.

To provide the location of each nozzle hole, use the Location sub-panel (see Reference Frames). Many required Spark Editor panel settings include:

1. **Timing**: Start and length of the spark event can be set with the option of defining a specific timeline in terms of crank angle or simulation time (see Time Frames).

2. **Spark Energy:**

   - **Energy Release Rate:** Configure the rate of energy release for the spark event.

   - **Energy Transfer Efficiency:** Set the energy transfer output from spark discharge into the gas mixture.

3. **Initial Kernel Radius:** Set the initial radius of the spark-ignition kernel.

3.4. Boundary Conditions Node.
Configuration of Boundaries for Automatic Mesh Generation.

The Boundary Node is used to describe the geometric surfaces. Because of the mesh configuration ANSYS Forte provides different choices for the Boundary Condition Node, depending on whether the network is a Body-fitted mesh or whether the automated mesh generation has been used. As explained above, the emphasis is on automatic mesh creation, so how to proceed is explained in the following point.

The following icons allow the numerous boundary setting options: New Inlet , New Outlet , and New Wall .

If a new boundary is formed, the first step is to connect the surface of the Geometry in the selection sub-panel of the location.

3.4.1. Inlet Panel

Inlet Panel is used to define gas conditions for the Inlet boundaries present in the geometry.

When ever a new inlet is needed, it can be created a new one using the **New Inlet** icon on the Boundary Conditions icon bar. The following icons also are use to modify the model: Rename , Copy , Paste , and Delete .

Inlet Editor Panel Options include:

1. **Composition**: To determine the composition specifics using the Mixture Editor . For details please see the Mixture Editor.

2. **Location**: Chose the location of the inlet in the location list.

3. **Inlet Options**: Choose Static Pressure, Total pressure, Velocity, or Mass Flow Rate from the list of inlet types, and each of these can also be Time Varying. Time Changing choices can involve an alternative timeline (see Time Frames).

   - If **Pressure Inlet** is chosen, for Time Varying cases, a constant pressure value or a pressure profile (as a function of time or crank angle) shall be provided.

- If **Velocity Inlet** is chosen, the magnitude and direction of the velocity must be defined, and there are options to determine Inflow Mixture Density. For low speed flows, the Assume Zero Pressure Gradient can be used as the norm.

- If **Mass Flow Rate Inlet** is chosen, the density of the input is calculated on the basis of the inner fluid cells adjacent to the inlet boundary. The direction of the velocity of the inflow can be either defined as Normal to Boundary or From Vector.

4. **Turbulence**: Pick a method for showing the inlet turbulence kinetic energy (TKE) and the dissipation rate of TKE from the chart. Options for Specification include:

   - **Turbulence Intensity and Length Scale**
   - **Turbulence Kinetic Energy and Dissipation Rate**
   - **Turbulence Intensity and Dissipation Rate**
   - **Turbulent Kinetic Energy and Length Scale**

5. **Temperature Option**: Specify the temperature behaviour as a defined static value, Assume Isentropic, or Absolute Temperature, each of which can also be Time Varying.

3.4.2. Outlet Panel

The Outlet Panel also present the following icons: Rename , Copy , Paste , and Delete , to modify an existing Outlet, where it can be copy and paste or elect the **New Outlet** icon on the Boundary Condition icon bar to create a new one.

Outlet Editor panel options include:

1. **Location**: Chose the location of the outlet in the location list.

2. **Outlet Options**: Select between the different outlet types: Static Pressure, Total Pressure, and either of these can also be Time Varying, or Continuative Outlet. If a Pressure Outlet is selected, an option is provided for Offset Distance to Apply Pressure. This is the distance of offset between the limit and the location of the specified fixed (ambient) pressure. For Pressure Outlets, a constant pressure value or a pressure profile must be entered, which varies as a function of time (or crank angle) (see Entering Profile Data).

3. **Turbulence**: Chose from the list a method, this parameter is used for reverse flow capability:

   - **Turbulence Intensity and Length Scale**
   - **Turbulence Kinetic Energy and Dissipation Rate**
   - **Turbulence Intensity and Dissipation Rate**
   - **Turbulent Kinetic Energy and Length Scale**

3.4.3. Wall Boundary

The Wall icon bar also offers 4 icons for managing walls: **Rename** , **Copy** , **Paste** , and **Delete** . A new wall can be created by copying and pasting an existing one or by using the **New Wall** icon on the Boundary Conditions icon bar.

The Wall Editor panel allows specification of the wall-boundary-layer model treatment option, the wall heat-transfer option, wall roughness parameters, and wall motion. For automated mesh generation every boundary surface of the wall may be described as a moving or non-moving wall independently.

1. Three option are used for the Wall-boundary-layer model treatment:

   - **Law of the Wall**: It is used in in internal combustion engines to model turbulent flows. It is available in both the turbulence models RANS and LES (see Turbulence)
   - **No Slip:** It is used with laminar flow simulation.
   - **Free Slip:** It is used fo special circumstances, when the wall boundary is at a far distance and does not affect the flows at all.

2. Wall heat transfer makes adiabatic inference, or assigns a fixed temperature value to determine the flow of wall heat. The temperature of the wall can differ constantly or spatially. Use the Profile Editor to create or specify the Temperature Table if variable.

3. Roughness of the wall causes pressure that can be important in turbulent wall-bound flows.

- The **Roughness Height:** By default it is zero and corresponds to a smooth wall. As the height of roughness is less than that of mesh used on the board.

- The **roughness constant** depends on the level of roughness. Its default value (0.5) refers to the roughness of closely packed, smooth sand grains.

4. Motion type: **Slider Crank** or **Offset Table** can be selected for the specification of the motion type, Which are listed in sections below.

3.4.3.1. Slider Crank Motion

Slider Crank Motion may be chosen as a form of motion when going a wall motion boundary, typically for movement of cylinders. The following parameter defines the characteristics of piston motion.

1. **Stroke**: Vertical length of the piston from bottom-dead center to top-dead center.

2. **Connecting Rod Length**: This is the distance of the connections between the crank arm and the piston pin.

3. **Piston Offset**: It is a measurement used when the piston is not aligned vertically. See in **Figure 3.14: Piston-pin offset distance** , the distance *x* is the piston offset.

   a - arbitrary piston position

   b - position at its true TDC

   c - position at its true BDC

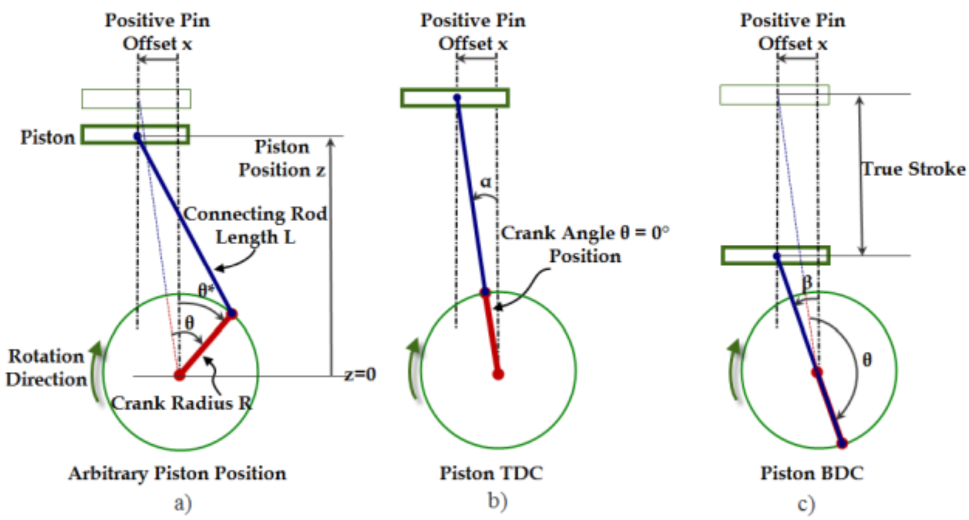

FIGURE 3.14: PISTON-PIN OFFSET DISTANCE

For any moving boundary the following parameter are needed :

- **Vertices to Transform**: Select **All** (appropriate, all the vertices will move together), or only **Interior** or only **Exterior**.
- **Direction**: A vector must be define in the desired direction with the most convenient coordinate system.

3.4.3.2. Offset Table Motion

An Offset Table Motion is another way to describe a Movement Function for a Moving boundary. Typically this alternative is chosen for type of valve motion. The following parameters define functionality of the movement:

1. **Vertices to Transform**: **All**, or only **Interior** (appropriate for valves, to allow only the **Interior** vertices to move) or only **Exterior** can be selected.
2. **Time Frame**: The time offset used to this wall.
3. **Direction**: A vector must be define in the desired direction with the most convenient coordinate system.
4. **Lift Profile**: Select or create a profile using the pull- down menu. (See Entering Profile Data , for details).

In **Figure 3.15: Intake valve Lift**, is shown the editor used to introduce an offset table motion.

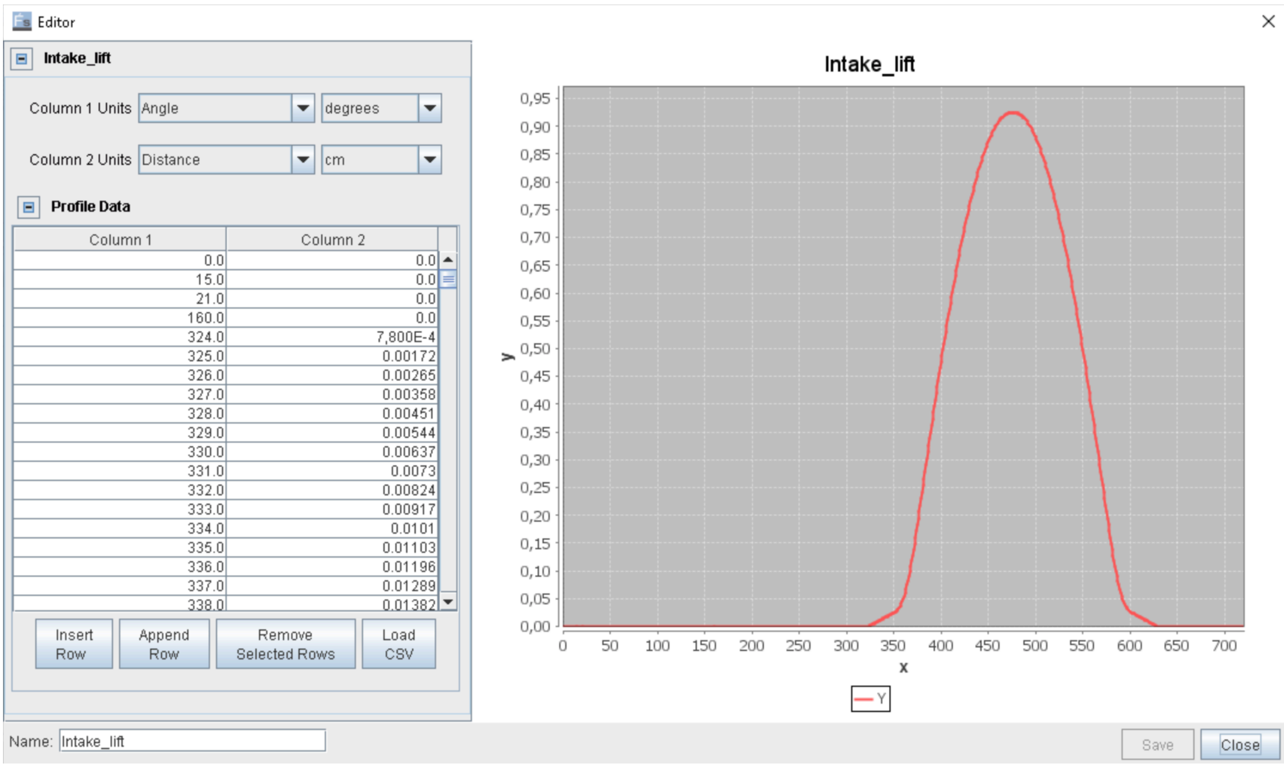

FIGURE 3.15: INTAKE VALVE LIFT.

3.4.3.3. Movement Type

Additional inputs are needed in case the moving boundary region is used to open or close a port. There are two forms of motion of the Valve and motion of the Sliding Interface. The operation of the valve is clarified in the next stage as Sliding Control operation is seen in situations involving 2-stroke engines.

For the Valve motion, the following features must be defined as the types of movement:

1. **Stationary Valve Seating Surface**: At zero lift pick the surfaces in contact with the valve.

2. **Valve Motion Activation Threshold**: Enter the minimum lift required for valve motion to be enabled. The valve does not open until the threshold is met by the valve lift profile.

3. **Approximate Cells in Gap At Min. Lift**: Set the minimum number of cells required in the gap upon valve lift (suggested range = 1.5 to 3 cells).

3.4.3.4. Valve-Seating Utility

As Valve Motion is chosen, if not already there is a utility available to shift the valve to the seated position. That is the closest angle between the vertices of the seated valve surface and its nearest points on the valve seat.

The utility uses this number to translate the valve to minimize the distance, the interior vertices of the valve are transformed and the geometry is updated.

Either of two items may be the product of this process: reaching the goal distance, the solver must consider a minimal distance greater than the resistance it needs.

3.5. Initial Conditions Node

This node is used to determine initial conditions for all variables of the structure, for each area specified in the geometry.

3.5.1. Configuration of Initialization Regions for Automatic-mesh Generation

A Default Initialization Node is automatically generated in the Initial Conditions tree thanks to automatic-mesh generation. In this node, the domain is initialized with the operating conditions, species concentration and specified temperatures.

In cases using automatic- mesh generation, there are two techniques for determining initial conditions:

1. To obtain specific initial conditions require the valves and/or sliding port interfaces to be separate regions.

2. To obtain specific initial requirements, let user-defined volumes get separate regions.

3.5.1.1. Strategy 1: Let the Valves and/or Sliding Port Interfaces Separate Regions

The Initial Conditions icon bar offers the option: **Secondary Region from Material Point**, to create a separate initialization element from the default one. If the new element is created, the Editor panel allows to customize the requirements and the initialization order must be set correctly.

The initialization order specifies which area in newly formed cells will be used to assign properties. In general the order would follow the course of the flow:

1. Intake port
2. Cylinder
3. Exhaust port

3.5.1.2. Strategy 2: Let User-defined Volumes Separate Regions

The Initial Conditions icon bar offers the option: **Secondary Region from Subvolume** , it is the same technique as the previous one, except it is now possible to choose a Subvolume instead of a floor.

3.5.2. Initialization Panel

Described as: Default, Cylinder, Port or a Subvolume region for the initialization objects. A determination of the temperature , pressure and composition of the species is required. Furthermore, kinetic energy turbulence, and components of velocity can be defined, if not zero.

1. **Composition**: To determine the composition specifics using the Mixture Editor. The Mixture Editor is used for a constant composition (see Mixture Editor for more details). Click the Pencil icon next to Create new ... for a composition which differs spatially. And the Table Editor for Initial Condition launches.

2. **Surface Composition and Dispersed Phase**: Use the Solid Phase Editor to determine the surface composition specifics and distributed phase properties. This is only needed if it is believed that particles are already present at simulation initialization.

The velocity can be specified as coordinate components in absolute values, or using a correlation of the swirl-flow. First pick Engine Swirl or General Swirl in the pull-down list next to Velocity in the Initialization Editor panel (Initial Conditions > Initialization) to determine swirl conditions

1. For the **Engine Swirl** option, set the following parameters:

   - **Initial Swirl Ratio**: That is the original measure of the angular velocity of the swirl flow proportional to the speed of rotation of the crankshaft (both at revolutions per minute)

   - **Initial Swirl Profile Factor**: This element is a dimensionless constant which determines the initial velocity profile of the azimuthal. It should have a value ranging from 0.0 (the wheel flow limit) to 3.83 (the wall velocity corresponding to zero). The suggested rating, and normal, is 3.11.

   - **Initialize Velocity Components Normal to Piston**: If verified, the axial velocity variable for an engine cylinder would be set to differ linearly from the piston surface to the top of the region (axial velocity = piston velocity on the piston surface, axial velocity = 0 at the top of the cylinder)

2. For **General Swirl**, set the **Reference Frame** (see Reference Frames) and the following parameters:

   - **Radius**

   - **Axial Velocity**

   - **Reference Angular Velocity**: The angular velocity defining the swirl velocity.

   - **Initial Swirl Profile Factor**: The initial swirl profile component, α, is a dimensional-less constant describing the initial azimuthal velocity profile and ranging from 0.0 (the wheel flow limit) to 3.83 (zero wall velocity). The suggested meaning is 3.11.

3.6. Simulation Controls Node

Simulation Controls Node allows to define the simulation time limits, RPM, time step, chemistry solver and transport terms. These are describe bellow.

3.6.1. Simulation Limits Panel

Simulation limits define the beginning and the end of the simulation in terms of time or crank angle.

1. **Time Based End Points**: Set the maximum simulation time.
2. **Crank Angle Based End Points**: Set the initial and end points for simulation.
   - **Initial Crank Angle**
   - **Final Simulation Crank Angle**
   - **RPM**: Engine Speed of the crankshaft rotation.
   - **Cycle Type**: specify a 4-stroke or 2-stroke engine.
   - **Engine Bore**: (Autumatic Mesh Generation only) Choose whether the engine bore should be calculated automatically or be user-specified.

3.6.2. Time Step Panel

ANSYS Forte simulations use advanced time-step adaptive control to allow the most effective solution while retaining accuracy during dynamic simulations where sharp gradients that occur in solution variables. Time-step is supervised by:

1. **Initial Simulation Time Step**: The first value of the time step.
2. **Restrict Time Step by Crank Angle**: This specifies a fixed average time-step size for engine instances, which is set by default to 1.1.
3. **Maximum Time Step Option**: The real transportation time phase in ANSYS Forte is calculated in an adaptive way, but the overall time phase constrains time-step changes that can support convergence during rapid change periods

4. **Advanced Time Step Control Options**: Within ANSYS Forte, the adaptive time-stepping algorithm changes the time-step size to maximize performance and precision in the solution.

3.6.3. Chemistry Solver Panel

Chemistry Solver Panel is used to describe tolerance to the solution. It is highly advised that the default chemistry solver choices be retained, as these have been checked to have the highest accuracy and speed. The options here are briefly listed below:

1. **Absolute Tolerance for Chemistry Solver**: Sets the absolute tolerance for the mass fractions of the species since they are solved with detailed kinetics. 1.0E-12 as default.

2. **Relative Tolerance for Chemistry Solver**: This alternative sets the relative tolerance for the mass fractions of the species as they are solved with detailed kinetics. 1.E-5 by default

3. **Use Dynamic Adaptive Chemistry**: This choice flips on an switch for the complex adaptive chemistry, it is off by default.

4. **Use Dynamic Cell Clustering**: This is another method that will significantly speed up the calculation by ensuring that in cells of the same thermochemical condition, chemistry computations are not excessively replicated.

5. **Use Autoignition-Induced Flame Propagation Model**: Enables the creation and propagation of flames that are caused by a cluster of self-ignited computer cells.

6. **Use Turbulence Kinetics Interaction Model**: Turns on the Model Turbulence Kinetics Interaction, providing a Mixing Time Coefficient alternative that regulates the turbulent mixing time scale. A greater importance applies to the influence of turbulent mixing on reaction speeds.

7. **Activate Chemistry**: select to have chemistry **Always On**, **Always Off**, or activated **Conditionally**.

8. **Disable Chemistry in the Unburned Region During Flame Propagation:** Selecting this alternative allows the measurement of the chemistry to be deactivated in the unburned area by using flame propagation.

3.6.4. Transport Terms Panel

The tolerances defined on the Transport Terms panel help to determine convergence for identified transportation variables. The defined tolerance values give the maximum relative error in the given variable's tacit solution. To control the accuracy of the transport solutions, the maximum number of iterations specified on this panel is used. If the stated limit is exceeded, then a reduced time stage will be used to replicate the measurement.

3.7. Output Controls Node

Output Controls Node is split in three items: Spatially Resolved, Spatially Averaged, Restar Data and Monitor Probes. It is explained in the points below, how to control the amount and the frequency of output for each type of data.

3.7.1. Spatially Resolved Panel

Spatially Resolved Panel must be activated checking on the **Spatially Resolved Output Control** box. Once the options are active, it allows to configure output based on **Time Steps**, based on **Crank Angles** (for engine simulations), or based on **Temporal** (time) values (for other types of simulations). Then the following features are display under the selected sub-panel:

1. **Crank Angle Output Control:**

    - **Interval Based Outputs. Output Every**: This helps determining the frequency in terms of the chosen units to save spatially resolved solution details.

    - **User Defined Crank Angle Output**: It can be selected a series of discrete points at which spatially resolved data will be saved, using the Profile Manager.

2. **Temporal Output Control:**

   - **Interval Based Outputs. Output Every**: This allows determining the frequency in terms of chosen time-related units for saving spatially solved solution details.

   - **User Defined Output (Time)**: A sequence of discrete periods can be picked, using the Profile Manager, when spatially resolved data will be preserved.

3.7.2. Spatially Averaged Panel

To unlock the Spatially Averaged Panel options, check the Spatially Average box to disable the **Spatially Averaged** Panel options. It has the same options as the previous panel except for the user-defined frequency. Limiting the frequency is not important, since the files take up far less space.

3.7.3. Restart Data Panel

Check the Reset Data box to trigger the **Restart Data** panel options. In a subsequent simulation, this panel allows the solution to be restarted from the point saved in the code

1. **Write Restart File at Last Simulation Step**: When the final simulation time is reached or the user finishes the simulation using the Run Simulation interface, the restart file will be written down.

2. **Interval Based Restart**: Write a reboot file at specified times, depending on the amount of fluid-time steps taken.

3. **User Defined Restart Points**: Useful discrete crank angles (for calculations on engines) or times to write restart scripts.

3.7.4. Monitor Probes Panel

To trigger the options click on the **New Monitor Probe** icon in the window. This panel allows parameters to be tracked at specified positions within a given time or length of the crank angle. The investigation has the following ways to monitor:

- Aspherical region.
- A boundary-condition-based region.
- An initialization region,.
- An injection region source,.
- An injection source intercepted by a plane.

3.8. Simulation Notes Panel

There are annotation methods for capturing any details that may be of assistance. The Bar icon for Simulation Notes has 1 symbol: **New Note**. To build the icon click on the button and then call the new page. The icons can be used in the Editor panel for copying, pasting, or removing the text, or capturing things about the simulation or creating a check index

3.9. Boundary Motion Panel

Until running the simulation, the Boundary Node Panel is used to display the required boundary motion in the 3-D View, so the errors can easily be seen as boundary interferences during the simulation.

On the Boundary Motion icon bar are located the tool to control the animation: **Play**, **Loop** , **Stop** , and **Reset.** See in **Figure 3. 15: Boundary Motion Panel.**

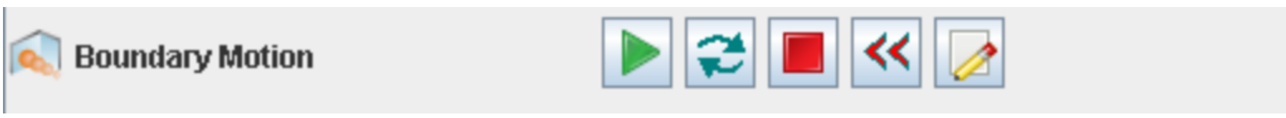

FIGURE 3. 15: BOUNDARY MOTION PANEL.

Users can preview any moment of the simulation by entering a crank angle or picking a point in a cycle plot.

Additionally, the Editor panel contains an optional view of plots es a rough map with different events and movements set up in the layout, such as boundary shift, ignition, or injection.

3.9.1. Mesh Generation Panel

When automatic-mesh generation has been used in the project, the Mesh Generation panel appears. This panel helps to preview the mesh in the simulation at different stages. Use the **New Automatic Mesh Plot** icon in the Mesh Generation icon bar to display the mesh which will be created at different times or crank-angles during simulation.

4. Job Execution

The nodes used for the setup and monitoring of the simulation are mentioned in this chapter. This is the finishing of the modeling process

4.1. Run Settings Node

The Run Settings node defines script preparation options, set MPI options, specify job-related directory names, and set environment variables, as well as advanced options.

The framework ANSYS Forte creates script files which are used to run the simulation(s). Choose to plan these scripts and perform a single simulation or conduct a analysis of parameters.

There is an alternative to schedule and pass the jobs on the local computer to another machine for batch submission or processing on a more efficient device.

4.1.1. Run Settings Panel

Forte Computing Options: Using certain options to monitor the options for production. To modify the output in the log file, select from three Logging Level options: Silent, Normal, or Verbose. For coding usually verbose is helpful.

4.1.2. Run Options

Job Script Options, it is used this sub-panel to fine-tune the development and execution of task scripts.

The **Default Run Typ**e option specifies the default value in the run simulation panel used for the Serial / Parallel buttons.

Multi-process jobs can run on different cores or machines, depending on the MPI implementation setup. When working with multiple processes a single job will finish faster.

The **Submit Action** option enables interactive or read-only activity. The default interactive mode ensures the work is automatically submitted on the screen where the user interface is running. Prepare Batch Scripts Either enable batch processing files to be loaded on a remote computing machine, or executed at a later date.

4.2. Run Simulation Node

The Run Simulation node is selected after the setup panels on the Workflow tree have been updated to describe the task. The Run Simulation products can be used for the application, tracking and management of run jobs

In the Run Simulation node the Run Settings overview is shown. The state of the simulation appears at the status column, it may be ready, run, stop, error or complete. The options for uploading and executing the script and another choice to avoid it are allowed. See in **Figure 4.1: Run Simulation node**, the displayed menu of the run simulation node.

When ANSYS Forte runs a simulation, there is an opportunity to monitor outcomes which will be clarified in the following paragraph

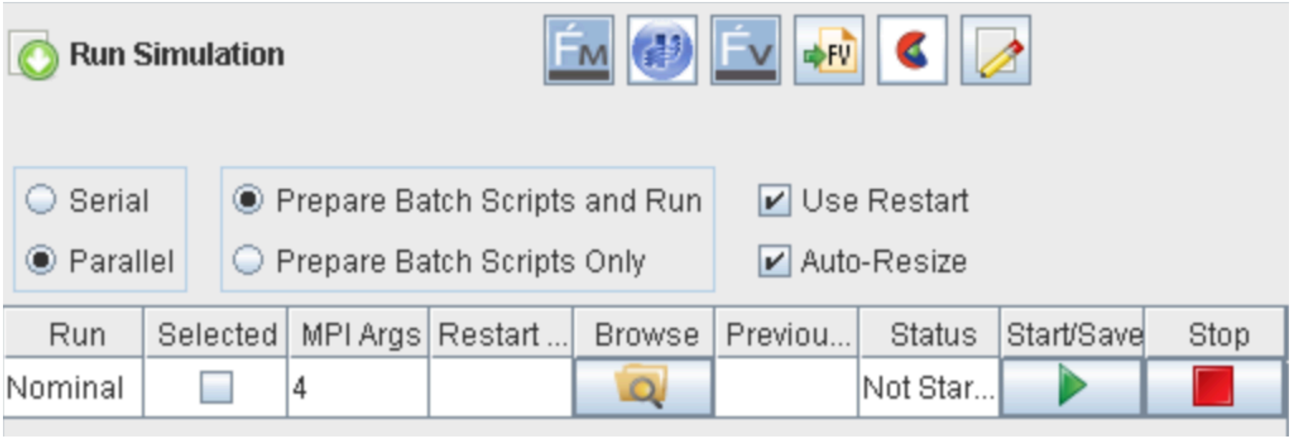

FIGURE 4.1: RUN SIMULATION NODE.

4.3. Monitoring Job Progress

While simulation jobs are running, results are being written into the solution file (.ftind and .ftres) as well as writing to several .csv files which can be used to plot advanced results.

Several comma-separated value (CSV) files are modified with plottable results for jobs that are running. It can be picked the alternative in the Run Simulation node. See in **Figure 4.2: Launcher dialog with Monitor Results icon highlighted**

FIGURE 4.2: LAUNCHER DIALOG WITH MONITOR RESULTS ICON HIGHLIGHTED

5. Solving an Indirect Injected Spark ignition Engine

This guide explains how to recreate combustion in an internal combustion engine with moving valves, using ANSYS FORTE in an indirect injection spark ignition. Engine geometry is imported, and the automated mesh creation from ANSYS Forte is used during simulation to generate the on-the-fly computational mesh.

5.1. Files Used in This Tutorial

The files for this tutorial are accessed by downloading the file siengine_pfi_automesh.zip on the ANSYS client portal, heading to the ANSYS client portal, going to http:/support.ansys.com/training. For this tutorial the files include:

1. ***SIEngine_Sample.stl*** : It is a file for geometry.
2. ***ExhaustLift.csv*** and ***IntakeLift.csv****:* Two data files detailing lifts for the intake and exhaust valves. Such profiles can be imported (from *.csv* files) or entered manually in the Profile Editor.
3. ***Spatial_Output_CAs.csv*** : Specifies when output of spatially solved data such as velocity, temperature, concentration of species etc.

5.2. Indirect-Injected Spark Ignition Engine

5.2.1. Problem Description

The use of gases as fuels is an issue for direct injected engines. In the intake port, right before the intake valves, the indirect injection is approximated by a pre-vaporized premixed mixture of fuel and air. Inside the combustion chamber there is no gas injector capable of dealing with the high pressure produced. This tutorial is performed for engine analysis and simulation of gases.

5.2.1.Import the Geometry

The Import Geometry option is available in the geometry node, go over there and press the **Import Geometry** icon. A drop down menu is then shown, Surfaces from STL file are chosen for this simulation. Find the required STL file, and accept the defaults, ***SIEngine_sample.stl.*** See in **Figure 5.1 Port-injected engine geometry with valves and ports defined,** all the surfaces generated in the model.

There are a number of actions with the imported geometry that can be performed on the Geometry node items, such as scale, rename, transform, invert normal elements, or delete geometry elements.

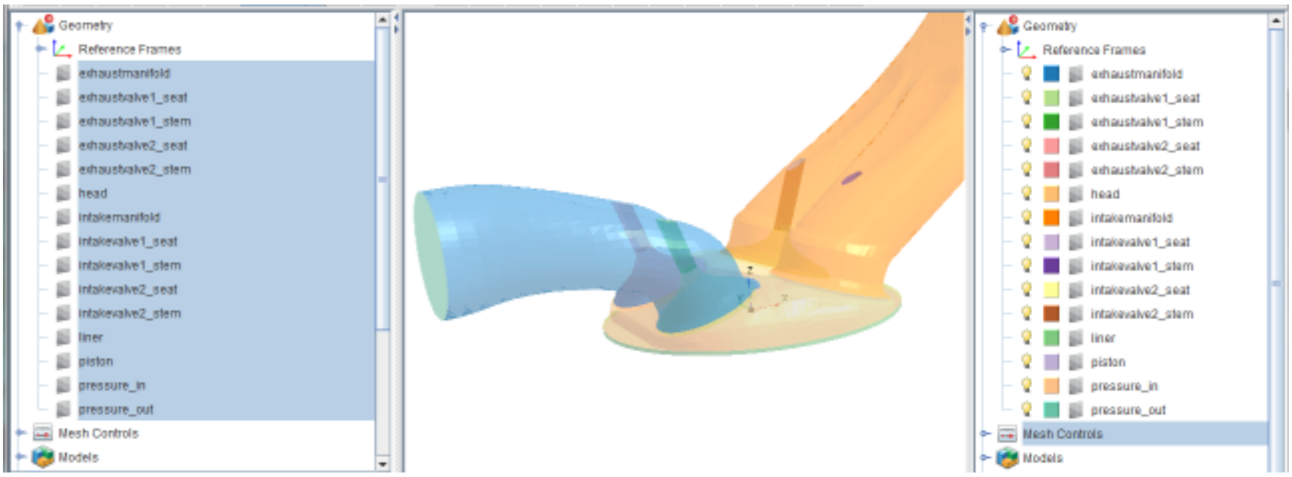

FIGURE 5.1: PORT-INJECTED ENGINE GEOMETRY WITH VALVES AND PORTS DEFINED

5.2.3. Sub-volume Creation

Go to the node Geometry, then select Sub-Volume and construct a sub-volume called **Chamber**. Select the surfaces below to describe the sub-volume: **Head, Liner** and **Piston**. See in **Figure 2.13. Chamber Selection**

Allow the default Reference Frame for the Material point using the Global Origin. Using Cartesian Coord to set Position. For X = 0, Y = 0, and Z = 1.0 cm.

Tap Apply.

5.2.4. Automatic Mesh Generation Setup

The next step in the Workflow tree is the Mesh Controls Node configuration. Material Point and Global mesh size specifically need to be defined. As explained before the Material Point, there should be at least one unit cell length away from any boundaries and the whole simulation must be within the domain.

1. In Mesh Controls, go to Material Point and use Global Origin to accept the default Reference Frame. Using Cartesian Coord to set Position. For X = 0, Y = 0, and Z = 1.0 cm. Apply.

2. Then go to Global Mesh Size to set the global **Mesh Size** to **0.2 cm**.

3. Finally set the **Small Feature Deactivation Factor** to **0.5.**

The following Table show how the mesh is refined for key geometric features.

Table 5.1: Mesh surfaces

Name	Location	Refinament Type	Size Fraction	Cell Layer	Active
Wall	All, except Pressure_In and Pressure_Out	Surface	1/2	1	Always
OpenBoundaries	Pressure_In and Pressure_Out	Surface	1/2	2	Always
ValveStems	IntakeValve1_Stem, IntakeValve2_Stem, ExhaustValve1_Stem, ExhasutValve2_Stem	Surface	1/4	2	Always
ValveSeats	Exhaust-Valve1_Seat, ExhaustValve2_Seat, IntakeValve1_Seat and IntakeValve2_Seat	Surface	1/4	1	Always
TDC1	Head, ExhaustValve1_Seat, ExhaustValve2_Seat, IntakeValve1_Seat, IntakeValve2_Seat, Liner and Piston	Surface	1/4	1	340-380 CA

Name	Location	Refinement Type	Size Fraction	Cell Layer	Active
TDC2	Head, ExhaustValve1_Seat, ExhaustValve2_Seat, IntakeValve1_Seat, IntakeValve2_Seat, Liner and Piston	Surface	1/4	1	690-790 CA
Spark	X = -0.36 cm, Y = 0 cm, Z = 0.33 cm	Point (radius = 0.5 cm)	1/4	-	680-720 CA

Adaptive Refinements: Adaptively refine temperature and velocity based on the solution variables.

1. Click the **Adaptive Meshing Solution** button, and call it SAM-Temperature. Set the Quantity Type = Gradient of Solution Field and Solution Variables = Temperature. Bounds Option = Statistical and Sigma Threshold = 0.5. Set the Size as fraction of Global Size = 1/4. The refinement is Active between Crank Angle 680 and 800 degrees, and the Location option is Sub-Volumes = Sub- volume-Cylinder. Click **Apply.**

2. Click the **Solution Adaptive Meshing** button and call it SAM-Velocity. Set the Quantity Type = Gradient of solution field and solution variable = VelocityMagnitude. Bounds Option = Statistical and Sigma Threshold = 0.5. Set the Refinement level = 1/2. The refinement is Active = Always, and the Location option is Entire Domain. Click **Apply.**

5.2.4. Models Setup

Once the mess has been specify it is time for the configuration of the Models Node. Follow the next steps for the configuration.

1. The import Chemistry icon allows to assign the chemistry. In this simulation the ***NaturalGas_1com_39sp.cks*** has been chooses as gas fuel, from the ANSYS Forte data directory. Then accept the Flame Speed Model defaults, except for Turbulent Flame Speed Ratio (b1) to 2.0 and click **Apply**

2. The default RNG k-e model turbulence setting are used. As it is explained in 3.3.4. Spark Ingnition, ANSYS recommend to keep the default Transport settings.

3. Select the Spark Ignition model and use the default parameters; 2.0 Kernel Flame to G-Equation Switch Constant, and 0.5 Flame Development Coefficient.

   Create New Spark on the New Spark icon.

   - In the Reference Frame, use Global Origin and Cartesian coordinates for the **Location**, and set X= -0.36 cm, Y=1.0E-6 cm, and Z=0.33 cm.

   - Set **Crank Angle for Timing** and Starting Angle = 688.0 degrees, Duration = 10.0 degrees.

   - For the **Spark Energy**, set Energy Release Rate = 20.0 J/sec. Accept the default 0.5 for Energy Transfer Efficiency and 0.5 mm for Initial Kernel Radius.

   Once the parameters for the New Spark has been specify click **Apply**.

4. Turn on the **Spray** model:

   4.1. Create a new **Solid Cone Injector** and name it Solid Inyector.

   - For **Composition**, select Create New… then select CH4 as the fuel species and specify the mass fraction as 1.0. Specify **Natural_Gas** as the name and click **Save**.

   - Set **Injection Type** to Pulsed Injection and Parcel Specification to Droplet Density with a Droplet Number Density of 1.

   - Set the **Inflow Droplet Temperature** to 400K.

   - For **Spray Initialization**, set Constant Discharge Coefficient and Angle and specify 0.7 for Discharge Coefficient and 14.0 Mean Cone Angle degrees.

- Select **Rosin-Rammler Distribution** for the Drop Size Distribution, use 3.5 for the Shape Parameter and specify the Initial Sauter Mean Diameter as 120 micron.

- Specify the following for the KH and RT model constants:
  - **Size Constant of KH Breakup = 0.5**
  - **Time Constant of KH Breakup = 10**
  - **Critical Mass Fraction for New Droplet Generation = 0.03**
  - Activate the **SMR Conservation in KH Breakup** option
  - **Size Constant of RT Breakup = 0.1 Time Constant of RT Breakup = 1.0 RT Distance Constant = 1.9**
  - Activate the **Use Gas Jet Model** and specify **0.5** for the **Gas Entrainment Constant**.

When all the parameters has been configured select **Apply**.

4.2. Add an Injection by clicking the **Injection**  icon and specify the following:

- **Injection Type** as Pulsed.
- **Timing** as Crank Angle. Specify the Start of injection as 420 degrees and the Duration as 18.4 degrees.
- **Velocity Profile**, select **Create New...**, the click the **Load CSV** button. Select the file called *injection_profile.csv*. And save it with the name of injection_profile.

  Select **Apply**.

4.3. Add the New Nozzle on the New Nozzle  icon and specify the following:

- **Location**, X = 8 mm, Y = 0 mm, and Z = 8.2 mm
- **Spray Direction** X =0 mm, Y = 0 mm, and Z = -2 mm.
- **Nozzle Size**, Diameter of 300 micron.

5.2.5. Boundary Conditions:

Boundary Conditions are specify for each geometry elements taking into account the jobs conditions of the surface.

1. **Inlet**: Click the **New Inlet** icon on the Boundary Conditions node to create it.
    - Pick **Pressure_In** from the Location list.
    - Choose **Total Pressure** as the Inlet Option with a constant pressure of 80000 Pa.
    - Select **Create New** in the composition dropdown list. Add Species of o2 =0.233 and n2= 0.767 and specify as a Mass Fraction. Finally save it as Air. See in **Figure 2.12: Creation of a gas mixture.**
    - Use **Turbulent Kinetic Energy and Length Scale of 10000 cm^2 /sec^2** and **1 cm.**

2. **Outlet:** Click the **New Outlet** icon.
    - Pick **Pressure_Out** from the Location list.
    - Select **Total Pressure** as the Inlet Option with a constant pressure of 100000 Pa, with an Offset Distance to Apply Pressure of 0.1 cm
    - Use **Turbulent Kinetic Energy and Length Scale of 10000 cm^2 /sec^2** and **1 cm.**

3. **Piston:** Click the **New Wall** icon.
    - Pick the **Piston** item in the Location list.
    - Select **Temperature Option** to **Constant** and **420 K**.
    - Turn **ON** the **Wall Motion** option and set the piston Motion Type to use a Slider-Crank Model with a Stroke of 7.95 cm and a Connecting Rod Length of 13.81 cm with 0.0 Piston Offset.
    - Choose the **Movement Type to Moving Surface** and accept the default Global Origin Reference Frame.

4. **Intake**: Click the **New Wall** icon. Choose the **IntakeManifold** item in the Location list and specify the **Temperature** to **300 K**.

5. **Exhaust**: **Intake**: Click the **New Wall** icon. Choose the **ExahustManifold** item in the Location list and specify the **Temperature** to **300 K**.

6. **Liner**: Click the **New Wall** icon. Pick the **Liner** item in the Location list and specify the **Temperature** to **385 K**.

7. **Head**: Click the **New Wall** icon. Pick the **Head** item in the Location list and specify the **Temperature** to **385 K**.

8. **Intake Valves**: Click the **New Wall** icon .

   - Pickform the Location list the **IntakeValve1_Seat, IntakeValve2_Seat, IntakeValve1_Stem** and **IntakeValve1_Stem** items.

   - Specify the **Temperature** to the constant value of **385 K**.

   - Turn ON (check) the **Wall Motion** and select the Motion Type to Offset Table.

   - For the Reference Frame accept the default Global Origin. Specify Spherical for the Coord. System under Direction for the valve motion and set θ=199 degrees and f=0.0 degrees.

   - Choose **Create New** from the **Lift Profile** drop- down list and click the **Pencil** icon. In the Profile Editor, click the **Load CSV** button and navigate to the *IntakeLift.csv* file. Name this IntakeValves. Set the units in the first column to Angle and the second column to cm. Save the Profile.

   - For Movement Type, change the pull-down menu to **Valve**. Then select the surface in contact with the valve.In this case, select both the IntakeManifold, the IntakeValve1_Seat and IntakeValve2_Seat.

   - Finally, set the **Valve Motion Activation Threshold** to **0.15 cm**.

   - In this case, accept the default value for **Approx. Cells in Gap at Min. Lift** of **3.0**.

9. **Exhaust Valves**: Click the **New Wall** icon .

   - Select form the Location list the **ExhaustValve1_Seat, ExhaustValve2_Seat, ExhaustValve1_Stem** and **ExhaustValve1_Stem** items.

   - Specify the **Temperature** to the constant value of 500 K.

   - Turn ON (check) the **Wall Motion** and set the Motion Type to Offset Table.

- For the **Reference Frame** Accept the default Global Origin f. Choose Spherical for the Coord. System under Direction for the valve motion and set θ= -199 degrees and f=0.0 degrees.

- Select **Create New** from the **Lift Profile** drop- down list and click the **Pencil** icon. In the Profile Editor, click the **Load CSV** button and navigate to the *ExhaustLift.csv* file. Name this ExhausValves. Set the units in the first column to Angle and the second column to cm. Save the Profile.

- For Movement Type, change the pull-down menu to **Valve**. Then select the surface in contact with the valve.In this case, select both the ExhaustManifold, the ExhaustValve1_Seat and ExhaustValve2_Seat.

- Finally, set the **Valve Motion Activation Threshold** to **0.15 cm**.

- In this case, accept the default value for **Approx. Cells in Gap at Min. Lift** of **3.0**.

5.2.6. Initialization:

The intake and exhaust must be initialized, assuming complete combustion, according to the limiting conditions values.

1. **Default Initialization:**

- Select 2 for the **Initialization Order**.

- Use **Composition Calculation.** Specify the Fuel Mass = 30 mg, select the fuel mixture in the Liquid input, for Air Flow set Phi = 1.0, EGR Fraction = 0.0, and for Internal EGR select the Estimate from CR and specify a CR of 10. Next, set the Calculate option to Exhaust and click Create Mixture. Specify a name of exhaust_gas. In the Initial Conditions, select exhaust_gas for the Composition.

- Specify a **Temperature** of **1,000 K** and the **Pressure** to **100,000 Pa**.

- Use the **Turbulent Kinetic Energy and Length Scale** option with values 10,000cm^2/sec^2 and 1cm for the Turbulence initialization.

- Set the **Velocity** to zero by using Velocity Components .

- Click **Apply.**

2. **Intake Initialization:**

- Choose the **New Port Initialization** icon and name it Intake.
- To identify the region, Set the coordinates X=6.0, Y=2.0, Z=5.0 cm, which is a point just inside the inlet.
- Set **1** as the **Initialization Order.**
- Select Air for the **Composition,** previously created and specify the Temperature to 300 K and Pressure to 80000 Pa.
- Use the **Turbulent Kinetic Energy and Length Scale** option with values 10,000cm^2/sec^2 and 1cm for the Turbulence initialization.
- Click **Apply**.

3. **Exhaust Initialization:**

- Choose the **New Port Initialization** icon and name it Exhaust.
- To identify the region, Set the coordinates X=-4.0, Y=2.0, Z=2.0 cm, which is a point just inside the exhaust.
- Set 3 for the **Initialization Order**.
- Select Exhaust for the **Composition**. Specify the Temperature to 650 K and Pressure to 100,000 Pa.
- Use the **Turbulent Kinetic Energy and Length Scale** option with values 10,000cm^2/sec^2 and 1cm for the Turbulence initialization.
- Click **Apply**.

5.2.7. Simulation Controls:

Simulation controls require the simulation limits, time stages, chemistry solver and transport conditions to be established.

1. **Simulation Limits**: Choose a **Crank Angle**-based simulation from a CA of **0 to 720** degrees. Set **RPM = 2,000 rpm**. The engine **Cycle Type** is **4-Stroke**.

2. **Time Step**:

    - Initial Simulation Time Step to 5.0E-7 seconds
    - Select Restrict Time Step by Crank Angle
    - Max. Crank Angle Delta Per Time Step of 1.1 degrees
    - Set the Max. Time Step Option to Constant and set the value of Max. Simulation Time Step to 1.E-5 sec.

    Kept the defaults for The Advanced Time Step Control Options settings:

    - Time Step Growth Factor = 1.3
    - Fluid Acceleration Factor = 0.5
    - Rate of Strain Factor = 0.6
    - Convection factor = 0.2
    - Internal Energy Factor = 1.0
    - Max. Convection Subcycles = 8

3. **Chemistry solver**: Kept the defaults:

    - Absolute Tolerance = 1.0E-12
    - Relative Tolerance = 1.0E-5
    - Select **Dynamic Cell Clustering t**o take advantage of cell groups that have identical conditions. Use 2 features to introduce Dynamic Cell Clustering: 1) Max. Temperature Dispersion of 10 K and 2) a Max. Equilibrium Ratio Dispersion of 0.05.
    - Use **Activate Chemistry Conditionally**, and select When Temperature is Reached with Threshold Temperature 600 K and also select During Crank Angle Interval between 650 and 850 crank angle, to raise the pace of time to solution . This ensures that chemistry is active during the time that combustion is expected even if temperature does not rise above 600 K.
    - Click **Apply**.

4. **Transport terms:** As it is explained in **3.6.4. Transport Terms Panel.** Use the defaults:

5.2.8. Outputs Controls:

It is used to store the desired data for simulation display, and to construct charts, diagrams, and animation.

1. **Spatially Resolved**: Helps control when processing of data will occur.

    - Specify the **Crank Angle Output Control** to report every **10 degrees**.

    - **User Defined Output Control m**akes rises in data volume. By importing the *Spatial_Output_CAs.csv* file, which has a list of specific crank angles where spatially resolved output will occur.

    - For **Spatially Resolved Output,** Select the following species: h2o, no, no2, co, co2, o2, ic8h18 and n2.

2. **Spatially Averaged**: Requires monitoring the output of averaged values across the domain.

    - Speficy the **Crank Angle Output Control** to reporting every **1 degree**.

    - For **Spatially Averaged Output,** Select the following species: h2o, no, no2, co, co2, o2, ic8h18 and n2.

3. **Restart data**: specify certain Restart Points using a separate file.

    - Switch User Defined Restart Points on (check) and use the Profile Editor to create a Restart profile.

    - Create a new profile called RestartOutput for this purpose, and add a line with a CA value set to 650.

5.2.9. Preview Simulation:

It is used to test the mesh created, and display the profiles for the model boundary movement.

- Select the **New Automatic Mesh Plot** icon, a preview mesh is generated, name it Preview.
- Choose Crank Angle as the Time Option and set it to 720 CA.
- Specify the Normal by Y=1.0
- Click Apply.

The sample mesh is generated on the Generate Mesh icon, and the boundary motion graphs are shown.

5.2.10. Run Simulation:

It is the final stage of the simulation process, when the state shows **READY** press the green **START** button.

Users can track the tests by clicking on the icon on Monitor Runs

5.2.11. Results:

Use the CFD-Post or ANSYS Forte Visualize to view results by opening the results file, *Nominal.ftind*, once the simulation process is complete.

6. Conclusion

ANSYS Forte is a very sophisticated software, specialized on internal combustion engines, and it is required many hours of training and studying for the appropriate use and comprehension of the programe.

The first idea for this project it was to develop a model from cero, but due to the coronavirus situation the uses of ANSYS Forte was not possible. However, all the knowledge acquired is used in the develop the guide for beginners.

For future proyecta it is a desire to continue with the learning procedure to achieve the goal of developing a complete new model.

7. Bibliography

The references for this final project has been taken from the study of the following articles:

[1] ANSYS Inc. and ANSYS Europe, “ANSYS Forte User’s Guide”, ANSYS, Inc. Southpointe 2600 ANSYS Drive Canonsburg, PA 15317.

[2] ANSYS Inc. and ANSYS Europe, “ANSYS Forte Tutorial 2019”, ANSYS, Inc. Southpointe 2600 ANSYS Drive Canonsburg, PA 15317.